Copy No._____

Defence Research and Development Canada
Recherche et développement pour la défense Canada

DEFENCE R&D DÉFENSE

Extension and Validation of UNDEX Analysis Methods

L. Donahue
Martec Limited

Martec Limited
Suite 400, 1888 Brunswick St.
Halifax, NS
B3J 3J8

Contract Number: W7707-02-1307/001/HAL

Contract Scientific Authority: Dr. J. Slater, 403-544-4731
Contract Project Manager: Dr. M. J. Smith, 902-426-3100 x383

Defence R&D Canada – Atlantic

Contract Report
DRDC Atlantic CR 2004-269
December 2004

Canada

Report Documentation Page

Form Approved
OMB No. 0704-0188

1. REPORT DATE	2. REPORT TYPE	3. DATES COVERED
DEC 2004		**-**

4. TITLE AND SUBTITLE	5a. CONTRACT NUMBER
Extension and Validation of UNDEX Analysis Tools	5b. GRANT NUMBER
	5c. PROGRAM ELEMENT NUMBER
6. AUTHOR(S)	5d. PROJECT NUMBER
	5e. TASK NUMBER
	5f. WORK UNIT NUMBER
7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES) **Defence R&D Canada -Atlantic,PO Box 1012,Dartmouth, Nova Scotia,CA,B2Y 3Z7**	8. PERFORMING ORGANIZATION REPORT NUMBER
9. SPONSORING/MONITORING AGENCY NAME(S) AND ADDRESS(ES)	10. SPONSOR/MONITOR'S ACRONYM(S)
	11. SPONSOR/MONITOR'S REPORT NUMBER(S)

12. DISTRIBUTION/AVAILABILITY STATEMENT
Approved for public release; distribution unlimited

13. SUPPLEMENTARY NOTES
The original document contains color images.

14. ABSTRACT

Numerous features have been added to both the IFSAS and Trident codes. Both codes were integrated into a single solver for 3D fluid-structure interaction simulations. A multiple material model using a mixed-cell approach was implemented in the IFSAS code, and validated against experimental data. Adaptive grid schemes for underwater shock and bubble analysis, hydrostatic pressure and airwater/seafloor boundaries, underwater explosion profiles, and fluid-backed shapes were also implemented. The improvements made to the Trident program include upgrading the translators between Trident and USA/CFA Version 5, automating the clipping of the fluid element meshes at the waterline, creating a link to the IFSAS code in the UNDEX shell program, and an extension of capabilities for analysing the response of equipment mounts to UNDEX loading.

15. SUBJECT TERMS

16. SECURITY CLASSIFICATION OF:			17. LIMITATION OF ABSTRACT	18. NUMBER OF PAGES	19a. NAME OF RESPONSIBLE PERSON
a. REPORT	b. ABSTRACT	c. THIS PAGE			
unclassified	**unclassified**	**unclassified**		**102**	

This page intentionally left blank.

Extension and Validation of UNDEX Analysis Methods

L. Donahue
Martec Limited

Martec Limited
Suite 400, 1888 Brunswick Street
Halifax, NS
B3J 3J8

Contract number: W7707-02-1307/001/HAL

Contract Scientific Authority: Dr. J. Slater, (403) 544-4731
Contract Project Manager: Dr. M. J. Smith, (902) 426-3100

Defence R&D Canada – Atlantic

Contract Report

DRDC Atlantic CR 2004-269

December 2004

Author

L. Donahue

Approved by

for J. Slater

J. Slater

Contract Scientific Authority

Approved for release by

Kirk Foster

DRP Chair

Terms of Release: The information contained herein is proprietary to Her Majesty and is provided to the recipient on the understanding that it will be used for information and evaluation purposes only. Any commercial use including use for manufacture is prohibited. Release to third parties of this publication or information contained herein is prohibited without the prior written consent of Defence R&D Canada.

Abstract

Numerous features have been added to both the IFSAS and Trident codes. Both codes were integrated into a single solver for 3D fluid-structure interaction simulations. A multiple material model using a mixed-cell approach was implemented in the IFSAS code, and validated against experimental data. Adaptive grid schemes for underwater shock and bubble analysis, hydrostatic pressure and airwater/seafloor boundaries, underwater explosion profiles, and fluid-backed shapes were also implemented.

The improvements made to the Trident program include upgrading the translators between Trident and USA/CFA Version 5, automating the clipping of the fluid element meshes at the waterline, creating a link to the IFSAS code in the UNDEX shell program, and an extension of capabilities for analysing the response of equipment mounts to UNDEX loading.

Résumé

De noumbreuses caractéristiques ont été ajoutées aux codes IFSAS et Trident. Ces deux codes ont été intégrés à un résolveur de simulations tridimensionnelles de l'interaction fluide-structure. Un modèle à plusieurs matériaux utilisant une approche à cellules mixtes a été inséré dans le code IFSAS et validé au moyen de données expérimentales. Des schémas de grilles adaptatifs sur l'analyse des chocs sous-marins et des bulles sous-marines, la pression hydrostatique et les limites de la surface et du fond marin, les profils d'explosions sous-marines et les formes à support liquide ont aussi été mis en oeuvre.

Les améliorations apportées au programme Trident comprennent le perfectionnement des traducteurs entre le code Trident et l'USA/CFA Version 5, l'automatisation du découpage des maillages d'éléments fluides à la ligne de flottaison, la création d'un lien avec le code IFSAS dans l'utilitaire de développement UNDEX et l'amélioration des capacités d'analyse de la réaction des supports de matériel aux charges produites par les UNDEX.

This page intentionally left blank.

Executive summary

Introduction

For several years DRDC Atlantic and DRDC Suffield have been supporting the development of an in-house underwater explosion (UNDEX) analysis capability. In the past this work has been performed in conjunction with the UNDEX experimental program at DRDC Suffield. More recently, application of these tools to Naval Force Protection studies has become a priority.

Principal Results

Several improvements have been made to the IFSAS analysis code that better allow it to perform three-dimensional UNDEX simulations of naval platforms. This includes simulating the response of a platform to a shock wave produced by an underwater explosion; as well as simulating the behaviour of the gas bubble produced by an underwater explosion and its effect on a platform.

Significance of Results

The improvements to the IFSAS code allow better simulation of close-range, shallow-water underwater explosions and their effects on naval platforms. These types of simulations are important for determining the vulnerability of a naval asset in a Force Protection scenario.

Future Plans

Further development of the IFSAS code (now called Chinook) is being performed by Martec Limited, with support from the Atlantic Innovation Fund. Force Protection studies being performed at DRDC Atlantic in 2004 and 2005 will use of the IFSAS and Trident codes extensively.

Donahue, L. (2004). Extension and Validation of UNDEX Analysis Tools. DRDC Atlantic CR 2004-269. Defence R&D Canada – Atlantic.

Sommaire

Introduction

Depuis plusieurs années, RDDC Atlantique et RDDC Suffield contribuent à l'élaboration d'outils d'analyse internes des explosions sous-marines (UNDEX). Par le passé, les travaux ont été effectués conjointement avec le programme expérimental UNDEX à RDDC Suffield. Plus récemment, la priorité a été donnée à la mise en application de ces outils dans le cadre d'études sur la protection des Forces navales.

Principaux résultats

Plusieurs améliorations ont été apportées au code d'analyse IFSAS pour qu'il puisse effectuer plus facilement les simulations tridimensionnelles UNDEX des plates-formes navales, y compris la simulation de la réaction d'une plate-forme à une onde de choc produite par une explosion sous-marine ainsi que la simulation du comportement de la bulle produite par une explosion sous-marine et de ses effets sur une plate-forme.

Portée des résultats

Les améliorations apportées au code IFSAS permettent de mieux simuler les explosions sous-marines en eaux peu profondes de courte portée et leurs effets sur les plates-formes navales. Ces simulations sont importantes pour déterminer la vulnérabilité des ressources navales dans les scénarios de protection de la Force.

Recherches futures

Martec Limited, avec l'appui du Fonds d'innovation de l'Atlantique, travaille à l'amélioration du code IFSAS (maintenant appelé Chinook). Les études sur la protection de la Force menées à RDDC Atlantique en 2004, et qui se poursuivront en 2005, font largement appel aux codes IFSAS et Trident.

Donahue, L. (2004). Amélioration et validation des outils d'analyse UNDEX. RDDC Atlantique CR 2004-269. R & D pour la défense Canada – Atlantique.

Table of contents

List of figures

List of tables

MARTEC™

Smart Solutions for Engineering,
Science and Computing

Extension and Validation of UNDEX Analysis Methods

Martec Technical Report # TR-04-23

October 2004

Prepared for:

DRDC Atlantic
PO Box 1012
Dartmouth, NS
CANADA
B2Y 3Z7

Smart Solutions for Engineering,
Science & Computing

Martec Limited
1888 Brunswick Street, Suite 400
Halifax, Nova Scotia B3J 3J8 Canada

tel. 902.425.5101
fax. 902.421.1923
email. info@martec.com
www.martec.com

SIGNATURE PAGE

EXTENSION AND VALIDATION OF UNDEX ANALYSIS METHODS

Technical Report # TR-04-23 Rev 02
29 October 2004

Prepared by: Laura Donahue — **Date:** Oct. 29/04
Laura Donahue
Junior Research Engineer

Merv Norwood — **Date:** Nov 1/04
Merv Norwood
Senior Research Engineer

Reviewed by: Richard Link — **Date:** Oct 29/04
Rick Link
Senior Research Engineer

Approved by: D. Whitehouse — **Date:** Oct 29/04
Dave Whitehouse
Senior Research Engineer
Project Manager

ACCEPTANCE BY TECHNICAL AUTHORITY

☐ Accepted as is
☐ Accepted subject to change (Details to be provided in attachment.)
☐ Rejected

Technical Authority: _____ **Date:** _____
Malcolm Smith
Defence Scientist - DRDC Atlantic

EXECUTIVE SUMMARY

Numerous features have been added to both the IFSAS and Trident codes. Both codes were integrated into a single solver for 3D fluid-structure interaction simulations. A multiple material model using a mixed-cell approach was implemented in the IFSAS code, and validated against experimental data. Adaptive grid schemes for underwater shock and bubble analysis, hydrostatic pressure and air-water/seafloor boundaries, underwater explosion profiles, and fluid-backed shapes were also implemented.

The improvements made to the Trident program include upgrading the translators between Trident and USA/CFA Version 5, automating the clipping of the fluid element meshes at the waterline, creating a link to the IFSAS code in the UNDEX shell program, and an extension of capabilities for analysing the response of equipment mounts to UNDEX loading.

This page intentionally left blank.

TABLE OF CONTENTS

LIST OF FIGURES

LIST OF TABLES

1.0 INTRODUCTION

Through initial contracts with Combustion Dynamics Ltd. (CDL) and Martec Ltd., DRDC initiated the development of underwater explosion (UNDEX) modelling capabilities [1-6]. Under follow-on joint contracts with Martec Limited and CDL, additional capabilities of the IFSAS/Trident system were introduced which involved modelling of general 3D hull shell structures with both air- and water-backed conditions; accounting for 3D structural buoyancy; implementing 3D hull whipping response capability in Trident; and modifying the CFD grid formation to allow an adaptive grid scheme (AGS) [1,2,7]. Currently, the 2D CFD code can model both shock decay and the bubble motion. The latter can produce strong and damaging jets during bubble collapse.

In recent years the IFSAS code has undergone redevelopment to enable the modelling of more complex geometries, new pre- and post-processing environments, and user-defined computational models. Additional work is needed to complete the redevelopment for near-field UNDEX problems. For far-field UNDEX problems, Martec Limited has recently developed an UNDEX shell program which contains empirically-based and equivalent-beam methods, as well as a modelling interface to USA/CFA.

Under the current contract, numerous features have been added to both the IFSAS and Trident codes. Both codes were integrated into a single solver for 3D fluid-structure interaction simulations. A multiple material model using a mixed-cell approach was implemented in the IFSAS code, and validated against experimental data. Adaptive grid schemes for underwater shock and bubble analysis, hydrostatic pressure and air-water/seafloor boundaries, underwater explosion profiles, and fluid-backed shapes were also implemented.

The improvements made to the Trident program include upgrading the translators between Trident and USA/CFA Version 5, automating the clipping of the fluid element meshes at the waterline, creating a link to the IFSAS code in the UNDEX shell program, and an extension of capabilities for analysing the response of equipment mounts to UNDEX loading.

2.0 INTEGRATION OF TRIDENT AND IFSAS

Martec's Trident implicit finite element code has been integrated with the IFSAS II CFD code into a single solver in order to perform underwater explosion analyses involving fluid-structure interactions. The IFSAS solver is used to calculate the blast loads due to an underwater explosion, while the Trident solver calculates the structural response due to the blast.

These two previously independent programs were compiled and linked together as a single executable. This involved some code alterations as the Trident solver is written in the FORTRAN programming language while IFSAS is written in C. To link the two programs, a single function was created to communicate between the modules.

One difficulty which was encountered during development regarded units. The CPF model had length units of millimetres, while IFSAS required input in metres. This problem was solved by creating a file (*units.tab)* which contains the length and pressure conversion factors required to convert the structural model into IFSAS units. This file gets called once, during initialization, as illustrated in Figure 1. A neutral geometry file (NGF) of the structure geometry is created by Trident. The IFSAS module is called and the *units.tab* file containing the length and pressure conversions is read. The NGF file is converted into IFSAS units, and any spaces in nodal numbering are replaced by dummy nodes which are not included in the connectivity matrix. These nodes are necessary because IFSAS requires nodes to be numbered consecutively.

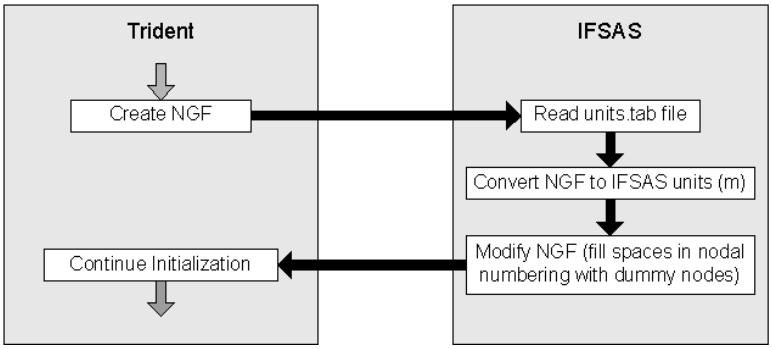


Figure 1: Communication Between Trident and IFSAS Modules During Simulation Initialization

With the new integrated solver, Trident is responsible for invoking and transferring the nodal velocities to the IFSAS module. IFSAS incorporates these velocities into its analysis, and calculates the resulting fluid field. IFSAS then transfers the pressures acting on the structural elements to the Trident module. This cycle is repeated every time interval, as specified by the user. The complete cycle is illustrated in Figure 2.

The fluid-structure coupling capability was tested for 3D problems, however no significant amount of work or testing was performed with 2D analyses.

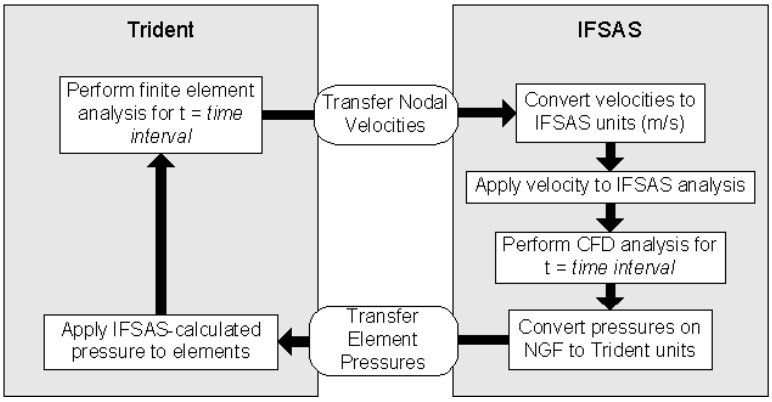


Figure 2: Communication Between Trident and IFSAS Modules During Simulation

3.0 UNDEX ANALYSIS CAPABILITIES FOR IFSAS-II

A number of capabilities have been added to the IFSAS II CFD code to enable underwater shock and bubble modelling. A near-field multiple material model was developed for simulating multi-phase flows which uses a mixed cell approach. This model is used in combination with adaptive grid schemes (AGS), hydrostatic pressure, and air-water and seafloor interfaces. Underwater profiles were created to speed up computation times.

3.1 NEAR-FIELD MULTIPLE MATERIAL MODEL

In order to provide an accurate numerical simulation of the bubble jet problem, the following must be considered:

1) CFD solver
2) Equations of state
3) Detonation products – water interface

These topics are discussed in the following sections, which were taken from the 2002 SAVIAC paper written on the subject (Appendix A).

3.1.1 Solver

An inviscid CFD solver is adopted, using the flux corrected transport (FCT) algorithm adopted by Boris [8]. The 2D axisymmetric equations of mass and momentum (Equations 1 and 2) are solved to obtain material densities, ρ, pressures, p, and radial and horizontal velocities, u and v , respectively:

$$\frac{\partial \rho}{\partial t} + \frac{1}{r}\frac{\partial (r\rho u)}{\partial r} + \frac{\partial (\rho v)}{\partial z} = 0 \tag{1}$$

$$\frac{\partial (\rho u)}{\partial t} + \frac{1}{r}\frac{\partial (r\rho uu)}{\partial r} - \frac{\partial (\rho uv)}{\partial z} = -\frac{\partial p}{\partial r} \tag{2a}$$

$$\frac{\partial (\rho v)}{\partial t} + \frac{1}{r}\frac{\partial (r\rho uv)}{\partial r} - \frac{\partial (\rho vv)}{\partial z} = -\frac{\partial p}{\partial z} \tag{2b}$$

In this formulation, internal energy is not considered, because the material pressure is assumed to be a function of density only.

3.1.2 Equations of State

The gas bubble is assumed to consist of air, and expand isentopically according to Equation 3:

$$p = p_0 \left(\frac{\rho}{\rho_0} \right)^{\gamma} \tag{3}$$

where

p_0 = 1 atm
ρ_0 = 1.18 kg/m^3

γ = specific heat ratio = 1.4

A modified Tait equation of state is adopted to model the water:

$$p = B\left(\left(\frac{\rho}{\rho_0}\right)^n - 1\right) + p_0 \qquad \frac{\rho}{\rho_0} > 1 \tag{4}$$

$$p = p_0\left(\frac{\rho}{\rho_0}\right)^m \qquad \frac{\rho}{\rho_0} \le 1 \tag{5}$$

where

B = 3000 atm
ρ_0 = 1000 kg/ m^3
n = 7.15
m = 21494.0

The exponent, m, is selected so that the slopes of the pressure vs. density curves are equal at atmospheric pressure.

3.1.3 Detonation Products - Water Interface

Since the bubble jet problem is modeled using discrete computational cells, each cell can have more than one material residing inside of it. Numerous approaches have been proposed to simulate multi-material problems, including level set methods [9], and mixed cell methods [10]. A mixed cell method assuming equal pressure in all cell constituent materials is adopted here. A mixed density, ρ_{mix}, is used that is a weighted average of the material volume fractions, φ_i:

$$\rho_{mix} = \rho_g \phi_g + \rho_w \phi_w \tag{6}$$

The subscripts g, w are gas and water constituents, respectively. Conservation of volume dictates that:

$$\phi_g + \phi_w = 1 \tag{7}$$

To obtain the volume fractions, and to maintain a conservative form of the equations, mass fraction tracers, T, are used, defined such that:

$$\rho_{mix} T_g = \rho_g \, \phi_g \tag{8}$$

$$\rho_{mix} T_w = \rho_w \, \phi_w \tag{9}$$

and conservation of mass (Equations 6, 8, and 9) dictates that:

$$T_g + T_w = 1 \tag{10}$$

Two extra equations are added to Equations 1 and 2 in order to determine the volume fractions:

$$\frac{\partial(\rho T_i)}{\partial t} + u\frac{\partial(\rho T_i)}{\partial x} = 0 \tag{11}$$

One equation is written each for gas and water.

It remains to determine the pressure in the mixed cell. The equilibrium pressure is determined by using an iterative process; the strategy employed here is as follows:

1) Solve Equations 1, 2, and 11 using FCT to obtain ρ_{mix}, T_g, and T_w.
2) Obtain lower and upper brackets for pressure, p_l and p_u, respectively, shown in Figure 3. These are obtained by assuming a pressure on either side, computing the constituent densities from the equations of state (Equations 3, 4, and 5), computing volume fractions from Equations 8 and 9, and computing the assumed mixed density from Equation 6. In this study, a bracket spacing of a factor of 10 was used between lower and upper pressures.
3) Once bracketed, use false position iteration to solve for the desired level of accuracy on ρ_{mix} shown in Figure 3. This is done by linearly interpolating left and right pressures to obtain a pressure estimate, p_r. From the pressure estimate, a similar procedure to Step 2 is employed to compute an estimate of ρ_{mix}. The bracket, whether left or right, is adjusted, and the iteration proceeds until the desired tolerance level is attained.

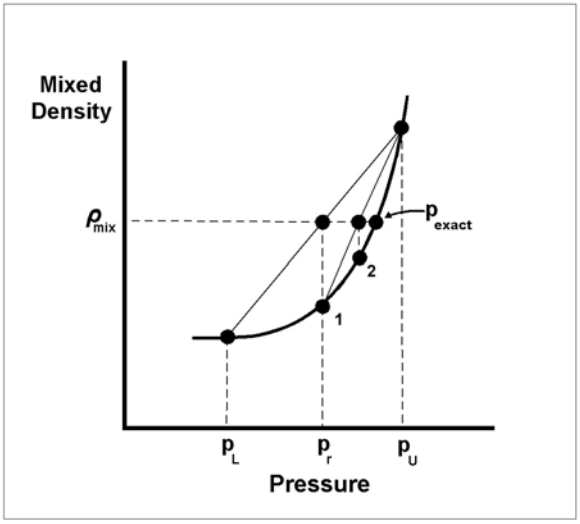


Figure 3: Iterative Method for Pressure Determination

3.2 ADAPTIVE GRID SCHEMES (AGS) FOR UNDERWATER SHOCK AND BUBBLE ANALYSIS

Adaptive isotropic, Cartesian grid refinement capabilities implemented in the IFSAS II code facilitated the modelling of underwater shocks and bubbles. Specifically, dynamic cell refinement was used. Using dynamic cell refinement, the refined area adapts to a specific local gradient, and therefore shifts during the simulation. AGS allows for coarser grids to be used, as regions of interest, such as shocks, can still be captured with a higher level of accuracy. This being said, the use of AGS still results in an increase in the overall computation time as the calculation time step is dictated by the smallest cell size.

Section 3.2.1 details how cells are refined, and Section 3.2.2 outlines the various available dynamic AGS criteria. Finally, a ship example using AGS is discussed in Section 3.2.3.

3.2.1 Refinement Levels

In IFSAS, the degree to which cells are refined is defined by the level of refinement entered by the user when specifying the AGS criteria. Figure 4 illustrates how cells are refined, using a 2D example. The largest cells are the root cells whose size is determined by the domain resolution. If an AGS with a maximum of level 0 refinement is specified, the root cells are not refined and there is no AGS refinement performed, even if an AGS criteria has been defined.

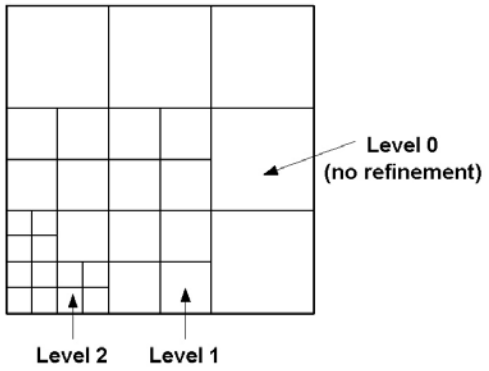


Figure 4: AGS Refinement Levels

With AGS Level 1, the root cells (or parent cells), are refined once. Accordingly, the smallest cells in a Level 1 refinement have twice the resolution of the domain. In Level 2 refinement the smallest cells have four times the resolution of the domain, and so forth. The number of refined cells (or children cells), n_c, produced from the refinement of a single parent cell is dependent on the problem dimensionality:

$$n_c = 2^d \tag{12}$$

where d is 1, 2, or 3 for one-dimensional, two-dimensional, and three-dimensional problems, respectively. Cells are refined and coarsened based on criteria to limit the total number of cells.

3.2.2 Dynamic AGS Criteria

The two primary criteria by which a grid can be dynamically refined are pressure gradient and density gradient. Following are descriptions of each.

Pressure Gradient
In pressure gradient-based refinement, cells are refined where there is a sufficiently large pressure gradient. The range of gradients for which cells are refined is specified by the user. This method is useful for capturing the pressure discontinuity across the normal shock of an expanding gas, shown in Figure 5 (left) for a basic shock tube [11]. Also, cells around solid surfaces are refined when this criteria is defined.

Density Gradient
Density gradient refinement is very similar to pressure gradient refinement, however more shockwave information is captured. While there is a density discontinuity across the normal shock as in the pressure case, there is also a discontinuity at the contact surface [11], shown in Figure 5 (right). Also, cells around solid surfaces are refined when this criteria is defined.

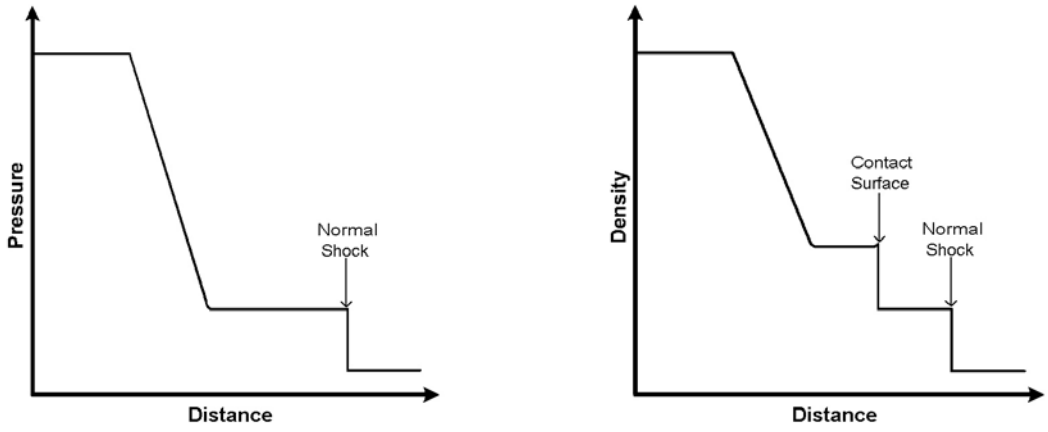


Figure 5: (left) Pressure Profile and (right) Density Profile for Shock Tube

3.2.3 3D Ship Example

A 3D AGS simulation was performed to test the capability. A TNT charge was detonated beneath a ship hull with the wetted surface illustrated in Figure 6. This geometry was imported into a 3D IFSAS simulation, and pressure gradient-based AGS was activated. Note that in this case a 3D FSI simulation was not performed; this simulation was executed using IFSAS II alone.

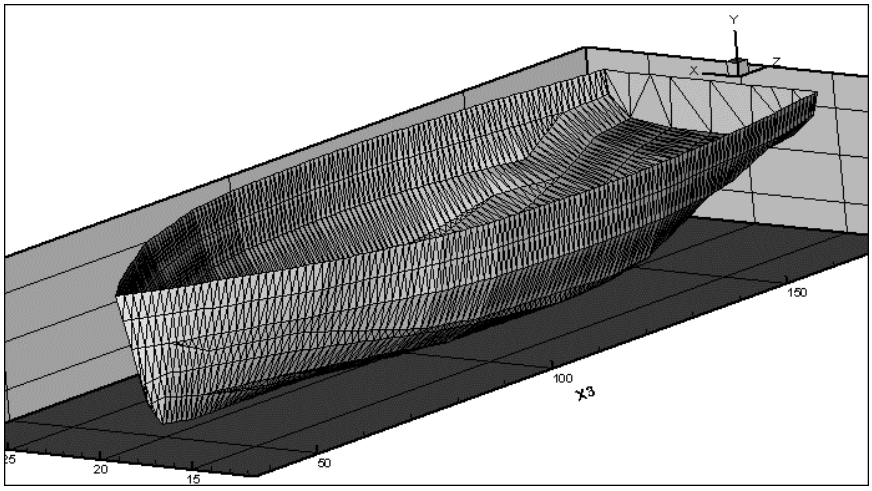


Figure 6: TNT Explosion Beneath Ship Hull with 3D AGS (X-Z Scales are not Proportional)

Figure 7 shows the results of the simulation for a cross-section along the width of the ship. The mesh at time = 0 ms is shown in Figure 7(a); here the refinement around the ship and the charge is clearly visible. As the explosion products expand (Figure 7 (b) to (d)), so too does the region of refinement.

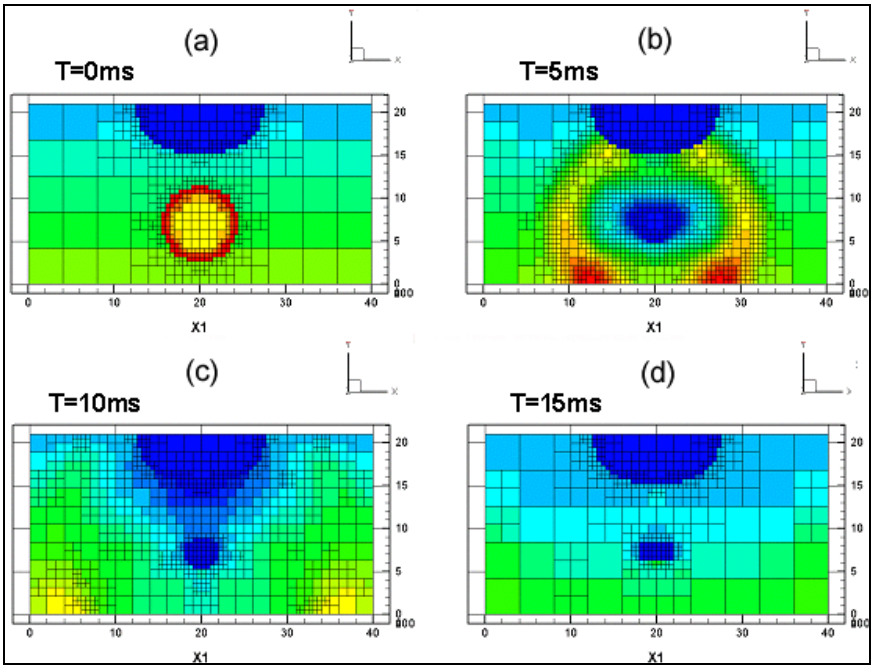


Figure 7: TNT Explosion Beneath Ship Hull with 3D AGS

3.3 HYDROSTATIC PRESSURE

Hydrostatic pressure was implemented in IFSAS for 2D and 3D calculations in order to simulate the effects of gravity on underwater explosions. Hydrostatic pressure allows the bubble surroundings at various depths of water to be simulated in greater detail. The hydrostatic effect was implemented as a source term on the vertical momentum equation equivalent to ρg, where g is the acceleration due to gravity. In addition, the initial hydrostatic pressure field is specified at the cell centres containing water, according to:

$$P = \rho g h \tag{13}$$

Figure 8 shows a representative calculation domain which has been initialized with hydrostatic pressure, which appears as horizontal lines of constant pressure.

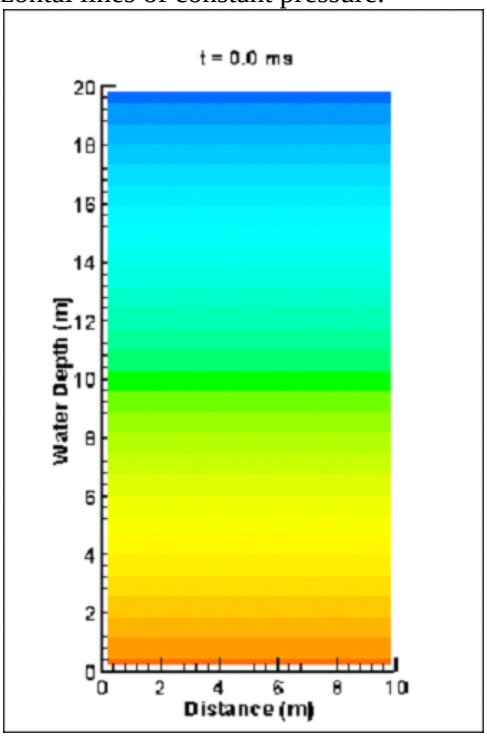


Figure 8: Initial Hydrostatic Pressure Field

3.4 AIR-WATER INTERFACE EFFECTS

An air-water interface was developed to model the interaction between a shockwave and the surface. This boundary condition predicts how a shock transmits through and reflects off of a free surface, without actually simulating the region of air above the surface of the water. The air-water interface is treated as a contact surface, which is solved using a Reimann solver.

A 1D test case was performed to verify the correct boundary behaviour. In this test case, a 10,000 atm Hugoniot shock in the water travels towards the air-water interface (located on the right-hand side of the domain). The left-hand boundary is an inflow with the Hugoniot condition (433 m/s). Figure 9 (left) shows the incident normal shock before collision with the free surface boundary. The wave transmission solution results in a reflected rarefaction wave in the water, as shown in Figure 9 (right). The transmitted normal shock into the air has been neglected. The reflected state behind the rarefaction is p = 1.244 MPa, ρ = 1000.524 kg/m^3, and U = 853.8 m/s.

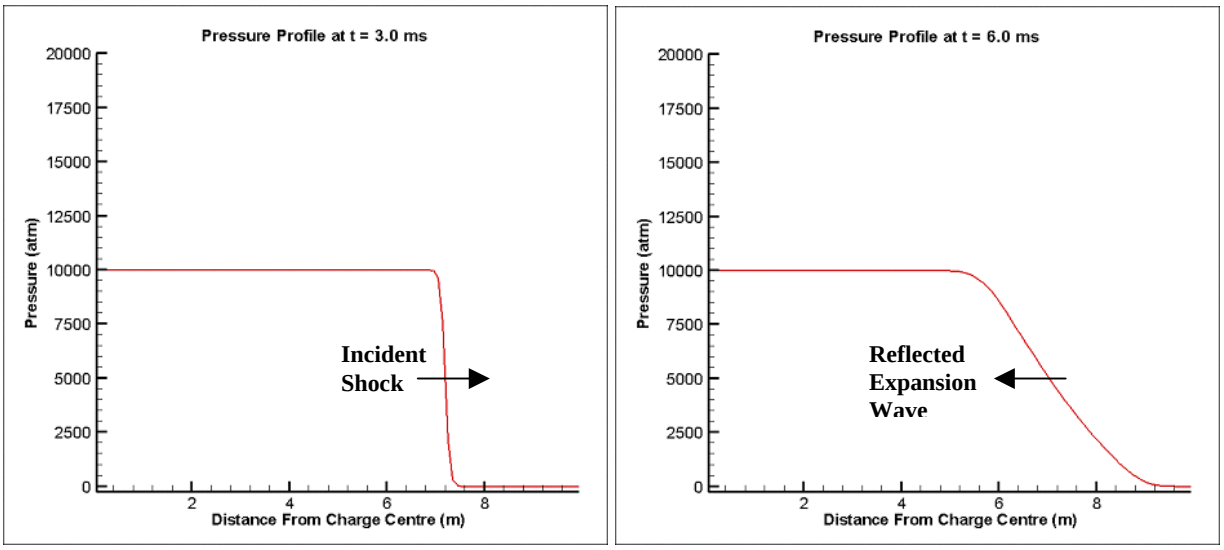


Figure 9: 1D Solution for Reflection from a Free Surface (left) Incident Shock (right) Reflected Expansion Wave

3.5 SEAFLOOR INTERFACE EFFECTS

A seafloor interface was developed to model the interaction between a shockwave and the seabed. The seafloor interface is treated as a contact surface, which is solved using a Reimann solver. The solid state is modelled as an elastic material with the properties of sand.

In this test case, a 10,000 atm Hugoniot shock in the water travels towards the seafloor interface (located on the right-hand side of the domain). The left-hand boundary is an inflow with the Hugoniot condition (433 m/s). Figure 10 (left) shows the incident normal shock before collision with the seafloor. The Hugoniot shock reflects from the interface as a normal shock with p = 1.925 GPa, ρ = 1321 kg/m^3, and U = 203.5 m/s.

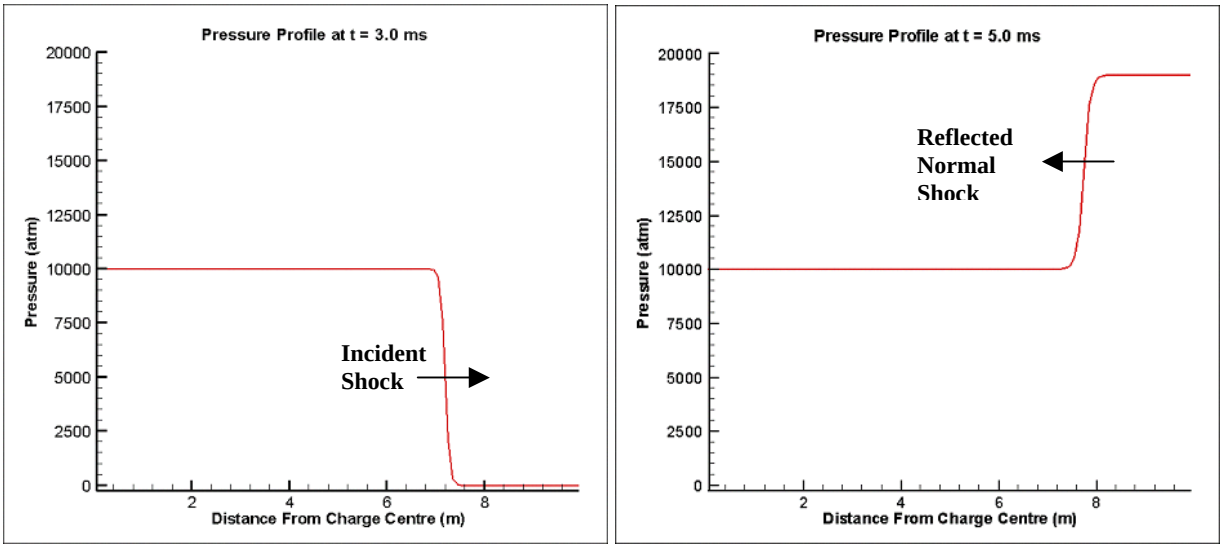


Figure 10: 1D Solution for Reflection from Seafloor (left) Incident Shock (right) Reflected Normal Shock

3.6 UNDERWATER EXPLOSION PROFILES

TNT profiles were implemented into the IFSAS code to improve the ease with which explosives can be modelled in multiple dimensions. They allow a wide range of explosives to be simulated without performing the calculations required for a balloon model each time the charge size or material is changed. Profiles are also useful in 3D problems when small increases in resolution come at the expense of long calculation times. Profiles give a high-resolution starting point for blast calculations which dramatically cuts down on the calculation time.

There are over 150 profile files available to IFSAS. These profile files were created from high resolution, one-dimensional calculations which used the balloon analogue model to simulate a 1 kg TNT charge. The density, pressure, and velocity profiles at varying shock wave positions along the domain were saved into individual files. This 1D profile data can then be mapped onto 2D circles and 3D spheres for multiple-dimensional calculations.

Scaling factors allow the 1 kg TNT profiles to be used for other explosives and charge sizes. The profiles are scaled for other explosives using the appropriate TNT equivalency factor. Table 1 gives TNT equivalency values for some other common explosives.

Table 1: TNT Equivalencies for Various Explosives [12]

Explosive	Pressure Equivalence	Impulse Equivalence
Composition C4	1.37	1.19
Composition B	1.11	0.98
HBX-1	1.17	1.16
RDX	1.14	1.09
Pentolite	1.42	1.00
Tritonal	1.07	0.96

When load duration, Δt, is low in comparison with the structure's natural period, T, the pressure impulse controls the structural response more than the shape of the pressure-time curve. Conversely, when $\Delta t/T$ is relatively large, the curve shape maters, and it is important to know the pressure magnitudes. In this case the pressure equivalence values should be used.

The published TNT equivalency values should be used with caution. While it may be appropriate to use these values for low-level calculations, or in the case that no experimental data is available, the equivalency values may need to be modified slightly to account for a particular set of initial conditions. If possible, preliminary calculations should be performed and compared with experiment to determine an appropriate equivalency value.

For different explosive radii, R, and charge masses, m_{charge}, IFSAS selects the profile corresponding to :

$$R_{1kg} = \frac{R}{m_{charge}^{1/3}} \tag{14}$$

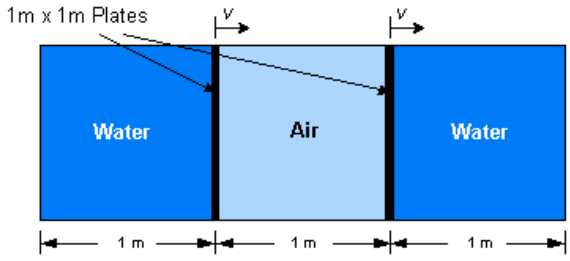
where R_{1kg} is the equivalent profile for 1kg TNT. Scaling in this manner has the effect of stretching or compressing the 1 kg TNT profile, depending on whether the charge size is larger or smaller. By investigating the pressure-time trace for a point in the domain, the scaling in Equation (14) is evident. The pressure-time traces are effectively scaled by:

$$t = t_{1kg}\ m_{charge}^{1/3} \tag{15}$$

where t_{lkg} is the timescale of the 1kg TNT profile, and t is the equivalent timescale for charge mass m_{charge}. With this change in the timescale, the impulse applied to a point is altered to account for the new charge size.

3.7 FLUID–BACKED SHAPES

Two simulations were created to test the ability of IFSAS II and Trident to model fluid-backed shapes. Figure 11 shows a schematic of the test cases. Two plates are moved at a constant velocity through the fluid domain. In the first case there is air between the plates, representing an empty hull compartment; in the second case there is water between the plates, representing a hull component filled with fuel or other liquid.



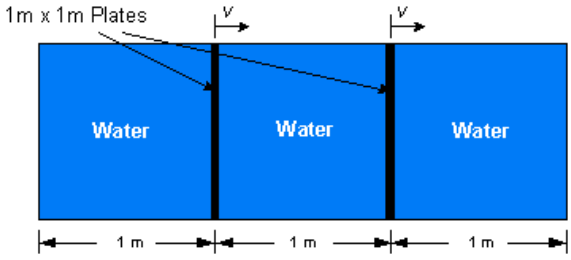


Figure 11: Fluid-Backed Shape Test Cases (top) Air-Backed (bottom) Water-Backed

Figure 12 shows the results of the air-backed case. As the plates travel through the fluid, the water pressure increases in front of the right plate. There is also a low pressure region trailing the left plate which is not evident in the figure. There is a negligible pressure increase between the plates due to the highly compressible nature of air.

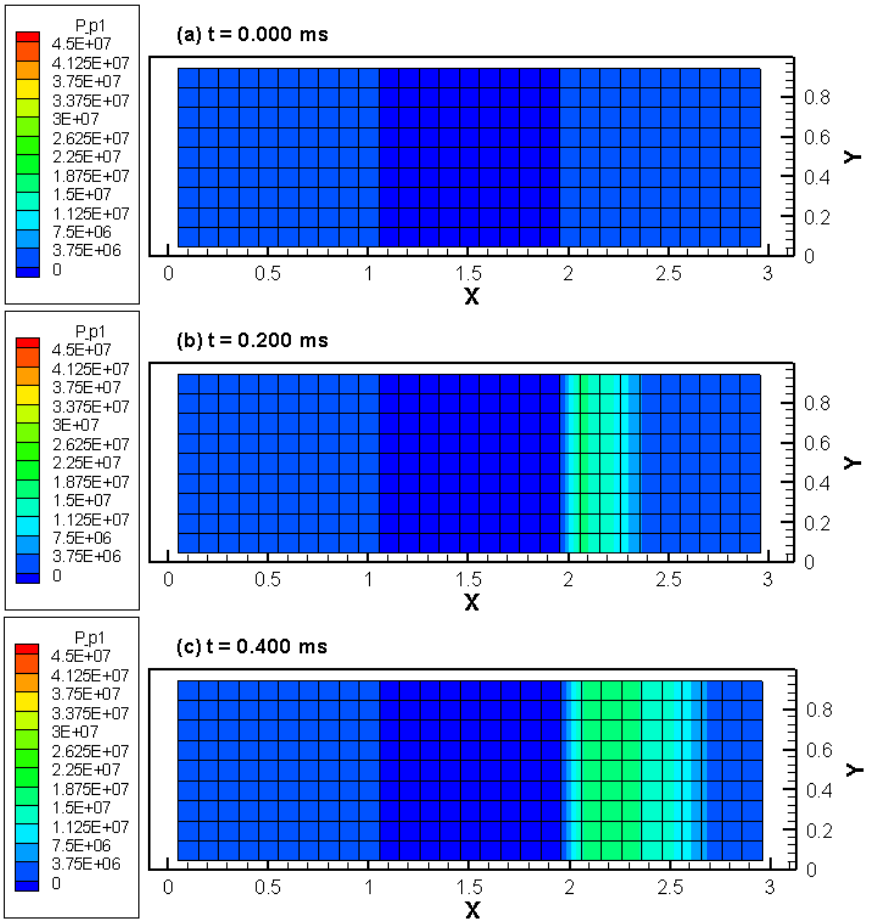


Figure 12: Simulation Results for Air-Backed Shapes

Figure 13 illustrates the results of the water-backed test case. In this case, the same pressure distribution is evident on the leading sides of both plates. As in the air-backed case, the low pressure region on the trailing edge is not apparent in the plots.

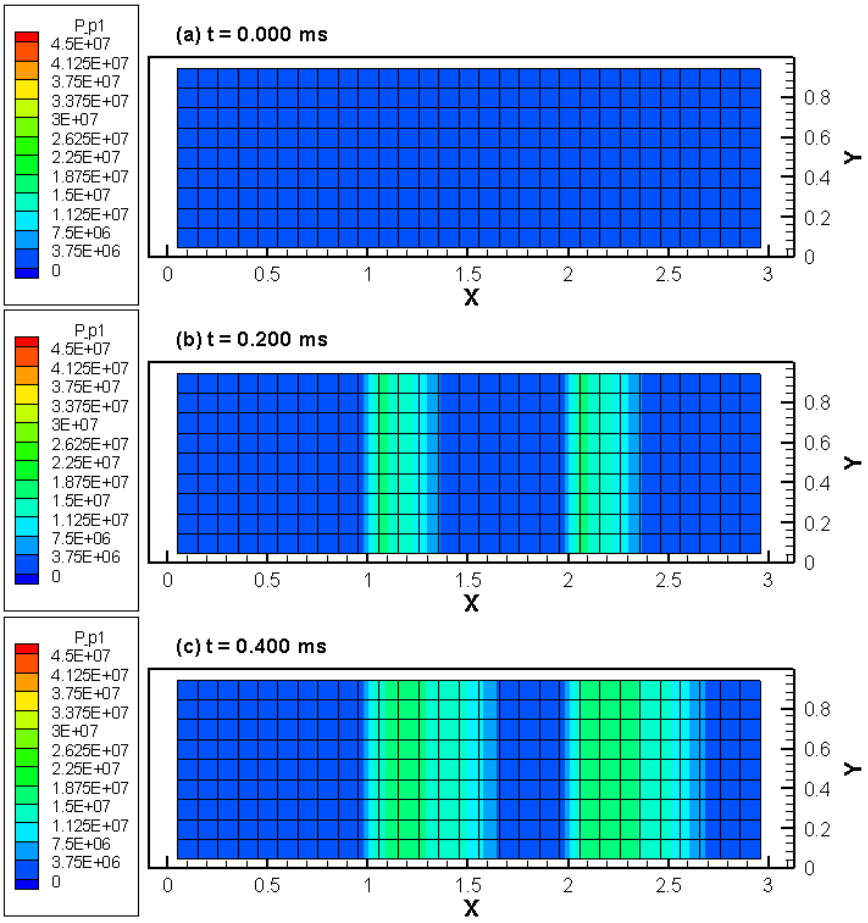


Figure 13: Simulation Results for Water-Backed Shapes

4.0 VERIFICATION AND VALIDATION

Some comparisons were made against experimental data in order to determine the effectiveness of the various implemented features. Bubble collapse and jetting phenomenon were simulated and compared with data from DRDC Suffield. Also, a verification exercise was performed to ensure that the IFSAS-Trident units conversion was correct.

4.1 BUBBLE COMPARISON WITH THEORETICAL

To demonstrate the UNDEX analysis methods, two problems are analyzed here; a free field bubble pulse, and a bubble jet on a non-responding target. For both problems, the layout is similar; 1 kg of TNT at 1000 m depth is employed. For the free field bubble, one-quarter of the explosive was considered, due to symmetry conditions. An axisymmetric computational grid was used, with an expansion grid used at the uppermost and right sides of the grid. The bubble energy was simulated by using a starting pressure in the gas bubble consistent with the ideal gas equation of state:

$$E = \frac{PV}{\gamma - 1} \tag{16}$$

where E is the total charge energy, P is the initial pressure, γ is the specific heat ratio of the bubble gas, and V is the gas bubble volume.

Figure 14 is a plot of the water mass fraction tracer and velocity vectors for four times, blue being gas, and red is water; one bubble pulse is shown. It can be seen that the pulse is consistent with observed experimental behaviour [13]. The bubble maximum radius and pulse time were calculated as 0.35 m and 7.0 ms, respectively. These compared well with similitude equations obtained from Keil [14] of 0.34 m and 6.6 ms.

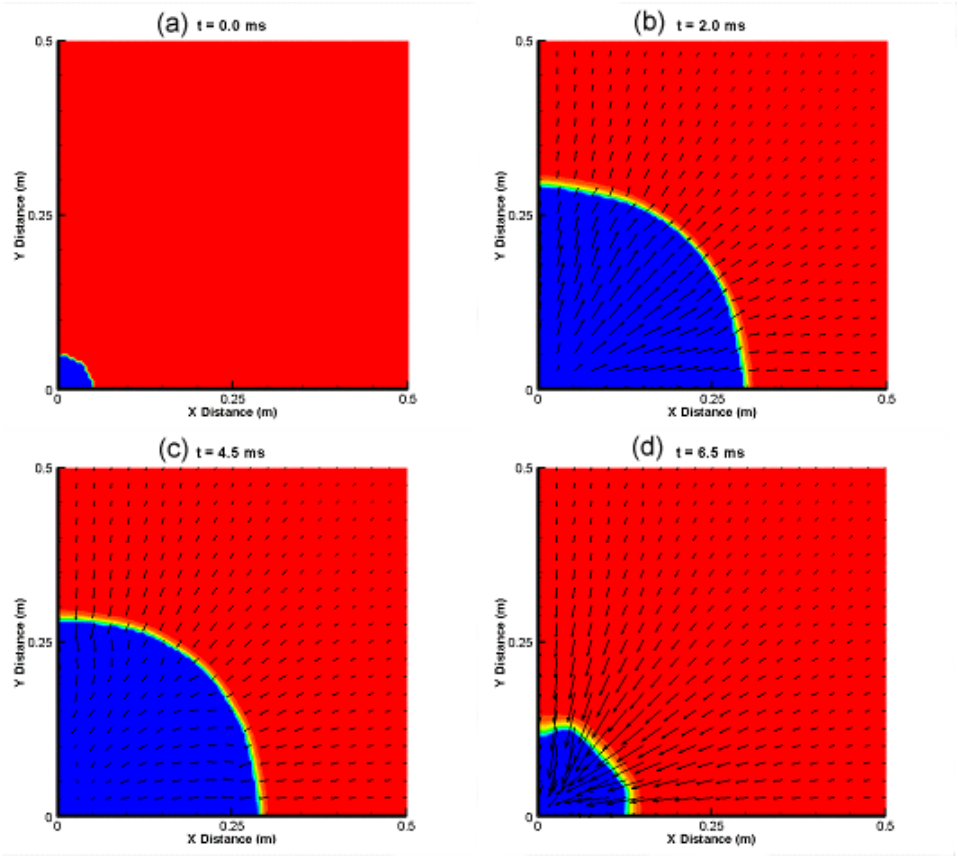


Figure 14: Mass Fraction and Vector Plot - Free-Field Bubble

An axisymmetric grid with expansion grids similar to those used in the bubble pulse problem were used for the bubble jet. Figure 15 shows the results of the jetting bubble for four times. The bubble initially undergoes a rapid expansion to a maximum radius, and flattens near the target. Eventually, the low pressure region re-pressurizes, and causes the formation of a high velocity water jet. The jet impinges on the bubble, and forms a toroidal shape which dissipates with increasing time. This is consistent with observed experimental results [13].

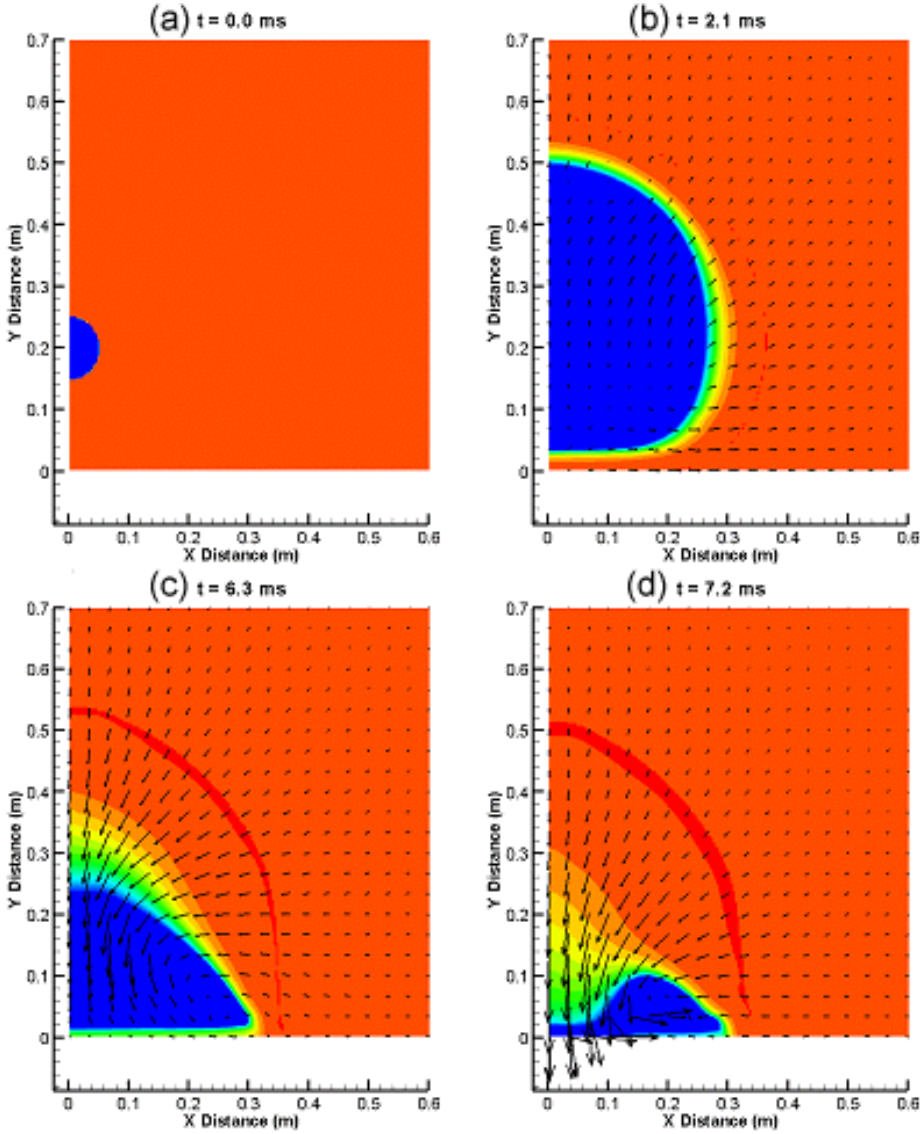


Figure 15: Mass Fraction and Vector Plot - Bubble Jet

Figure 16 shows a plot of pressure vs. time at the center of the non-responding target. The results show that there is a large initial pressure from the shockwave that has a relatively small duration. This is followed by a long low-pressure phase and a high pressure-high impulse phase from the jetting bubble. The jetting pressure is significantly longer than that developed by the initial shockwave.

The current simulations were performed without applying hydrostatic pressure. In these simulations, the hydrostatic pressure is huge compared with the gradient across the bubble as the simulated depth is on the order of 1000 m. It is anticipated that in this case, the effect of hydrostatic pressure is negligible. The effect of hydrostatic pressure on the bubble shape will be investigated in future studies of bubbles in closer proximity with the free surface.

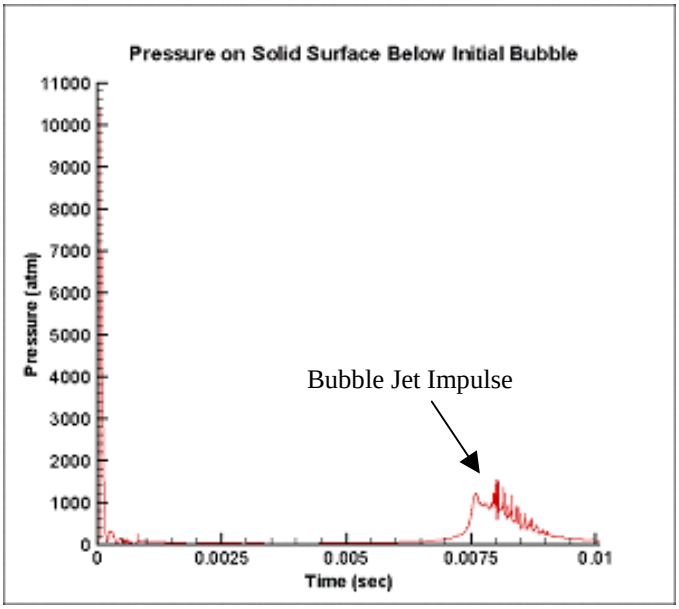


Figure 16: Pressure as a Function of Time for Bubble Jet Simulation

4.2 METRES AND MILLIMETRES VERIFICATION

One difficulty encountered during the current work was associated with model units. The CPF models in use have length units of millimetres, however IFSAS requires information in metres. Therefore, units must be converted during initialization and coupling. This is discussed in Section 2.0.

The units conversion method was verified by performing a simple test. Figure 17 illustrates the simulation configuration. A 1 kg TNT charge is placed below a steel plate submerged in water. The plate edges are fixed. The simulation was run twice. In the first case, the dimensions were specified in metres, and the conversion factors required by IFSAS for length and pressure were both 1.0. In the second case, the dimensions were specified in millimetres, and the conversion factors for length and pressure were 10^{-3} and 10^{-6}, respectively.

The absolute, resultant translation and rotation of selected nodes on the plate were monitored. The values extracted at 3 ms and 10 ms are shown in Table 1, with the plate deformation at these times, as well as the locations of the nodes being monitored, shown in Figure 18. As shown in the table, the translations are equivalent considering the length units in each case, and the rotations are equal as these are independent of model units.

The translations and rotations at points 17 and 18 are not equal, although these points may appear at first to be symmetrical. Both the finite element and the CFD meshes were relatively coarse in this test. Figure 17 shows the plate mesh; note that the relatively coarse triangular mesh results in symmetry plane perpendicular to the page. This plane of symmetry is different from the plane along the other diagonal based on the element orientations. If the results at node 17, for example, were compared with the results for the node diagonally opposite, there would be much better agreement. Also, the charge shape, as detected on the CFD mesh, may not have been symmetrical with respect to the centre of the plate, again due to the coarse resolution. With higher CFD and finite element mesh resolutions, the solution at node 17 should approach that at node 18.

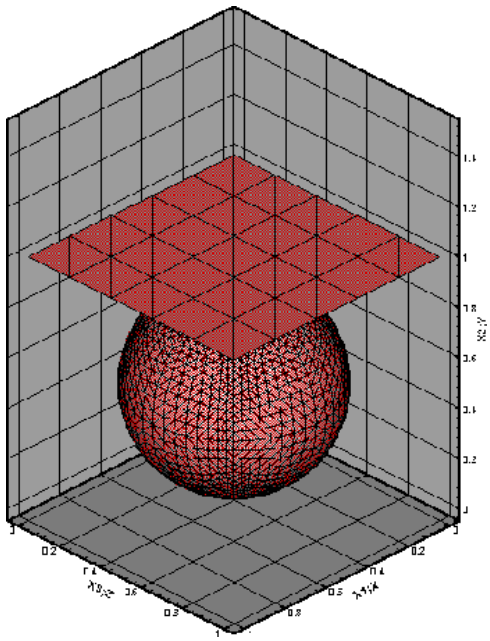


Figure 17: Simulation Configuration for Units Verification

Table 2: Comparison of Translations and Rotations at Select Nodes for Units Verification

node 31		T_{res}	R_{res}
t = .3e-2	m	1.4211e-02	1.9974e-01
	mm	1.4211e+01	1.9974e-01
t = .1e-1	m	1.4047e-01	1.4448e+00
	mm	1.4047e+02	1.4448e+00

node 12		T_{res}	R_{res}
t = .3e-2	m	1.7697e-01	1.8922e+00
	mm	1.7697e+02	1.8922e+00
t = .1e-1	m	3.7001e-01	4.8557e+00
	mm	3.7001e+02	4.8557e+00

node 17		T_{res}	R_{res}
t = .3e-2	m	6.8150e-01	8.3573e+00
	mm	6.8150e+02	8.3573e+00
t = .1e-1	m	9.5396e-01	6.6886e+00
	mm	9.5396e+02	6.6886e+00

node 18		T_{res}	R_{res}
t = .3e-2	m	8.1518e-01	6.4707e+00
	mm	8.1518e+02	6.4707e+00
t = .1e-1	m	1.0842e+00	4.8493e+00
	mm	1.0842e+03	4.8493e+00

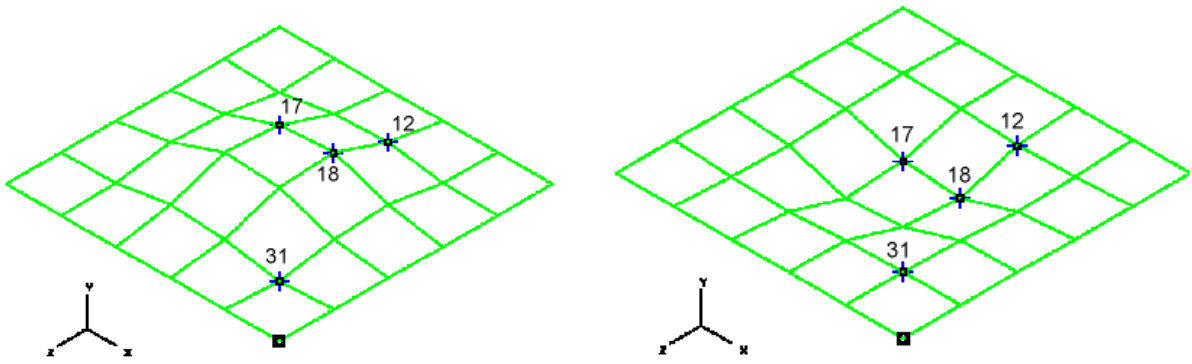


Figure 18: Deformed Plate at (left) t = 3 ms and (right) t = 10 ms

5.0 TRIDENT IMPROVEMENTS

5.1 UPGRADE TO TRIDENT-USA/CFA TRANSLATORS

During the period, two versions of the USA/CFA code were installed at the DRDC Atlantic site. The first version to be installed was USA5. After some minor modifications to the translators, the new version worked fine. However, due to some small changes in the USA/CFA input data, it was found that old models would not run. With fairly little effort, the necessary changes were identified, and the old models were then modified and successfully executed. The second version to be installed was USA6. Whereas USA5 uses integer*4 addressing, USA6 uses integer*8 addressing. The advantage in the latter is that much larger fluid models can be handled.

5.2 COUPLING NON-LINEAR TRIDENT-USA/CFA

The coupling of non-linear Trident to USA/CFA was investigated. It was established that the simplest approach to linking the two systems would be either using a predetermined time step that would be the same for both Trident and USA/CFA or allowing the codes to run at different time steps. Trident does permit a predetermined time step, with the possibility of updating it during a restart. Template source codes for developing such a link were obtained from the USA/CFA vender. Furthermore, the Trident to IFSAS II link, developed during this project and described in Section 2.0, is very similar to the one needed for the Trident to USA/CFA link. Particularly important and significant is the ability of the Trident and IFSAS II codes have to run using different time steps.

5.3 AUTOMATIC CLIPPING OF FLUID ELEMENTS

A capability to automatically (or transparently) clip fluid element models for added mass calculations has been added to the Trident system. When the fluid element model is used in the Trident solver for added mass calculations, only the portion of the model below the user-defined waterline is taken into account. A feature has been added for visually checking the waterline prior to a Trident added mass calculation. Also included in this task was the ability to use any of the three global coordinate planes for the waterline plane. This now being the case with the waterline plane, the ability to use any one of the three global coordinate planes for plane of symmetry had to be incorporated. Furthermore, a feature was added to Trident for visually checking the user-defined plane of symmetry.

5.4 LINK TO IFSAS CODE FROM UNDEX SHELL

Earlier, the old UNDEX Shell was incorporated directly into the Trident system as an Application Interface (API). At the same time the IFSAS component of the old UNDEX Shell was separated from the new UNDEX Shell and offered as a separate API called IFSAS. In addition to the previous feature for generating Neutral Geometry Files, the new IFSAS has a feature for modifying an existing IFSAS II input data file to include parameters unique to UNDEX. The FE Solver wizard in Trident has been updated to include the option create input files for a coupled Trident/IFSAS analysis. Both linear and non-linear analyses can be performed. The IFSAS II API is demonstrated in Appendices A, B and C.

5.5 EQUIPMENT MOUNTS EXTENSION TO TRIDENT

No work was completed regarding the equipment mounts extension to Trident. Instead, the resources planned for this task were re-allocated to the integration of Trident and IFSAS. This came about as the result of developing an unplanned single executable and the additional effort needed to accommodate the Fortran and C programming languages.

6.0 CONCLUSIONS AND RECOMMENDATIONS

Under the current contract, numerous features have been added to both the IFSAS and Trident codes. Both codes were integrated into a single solver for 3D fluid-structure interaction simulations. A multiple material model using a mixed-cell approach was implemented in the IFSAS code, and validated against experimental data. Adaptive grid schemes for underwater shock and bubble analysis, hydrostatic pressure and air-water/seafloor boundaries, underwater explosion profiles, and fluid-backed shapes were also implemented.

The improvements made to the Trident program include upgrading the translators between Trident and USA/CFA Version 5, automating the clipping of the fluid element meshes at the waterline, creating a link to the IFSAS code in the UNDEX shell program, and an extension of capabilities for analysing the response of equipment mounts to UNDEX loading.

There are some areas which may be addressed by future bubble model developments. The bubble model described produces results that are qualitatively consistent with observed experimental behaviour, however there were some observed problems with the computational scheme.

1) FCT Scheme: The FCT algorithm exhibited density oscillations in some problems that resulted in non-physical negative densities. This phenomena has been well documented in the CFD community [16]. A Godunov scheme should minimize or eliminate spurious negative densities if an appropriate multi-phase approach can be defined.

2) Diffusion: Since the interface was not explicitly tracked to within one cell resolution, several mixed cells resulted. Since air is significantly more compressible than water, the mixed material may be less stiff than what actually exists, and may result in lower pressures and impulses. This could be resolved by including a SLIC or variant interface tracker [17] to maintain a sharp interface.

3) Cavitation: Bubbles at shallower depths were attempted, however, it was found that the cavitation part of the Tait equation of state may have been too severe; at shallow depths, the bubbles never contracted once the maximum expansion was reached. A floor cavitation pressure similar to that used in Luton and Wardlaw [18] will be used to try and resolve this problem.

7.0 REFERENCES

[1] "Integrated Fluid Structure Interaction Software (IFSAS Version 1.6): Computational Fluid Dynamics (CFD) User's Manual", Combustion Dynamics Ltd. (CDL), 1997.

[2] "Integrated Fluid Structure Interaction Software (IFSAS Version 1.6): Computational Fluid Dynamics (CFD Reference Manual", Combustion Dynamics Ltd. (CDL), 1997.

[3] Tran, D., Link, R., Penrose, J. and Thibault, P.A., "CFD Modelling of Fluid-Structure and Liquid-Gas Interactions", 2nd Cdn CFD Conf., UTIAS, Toronto, ON, pp 171-178, July 1994.

[4] Slater, J.E. and Norwood, M.E., "Naval Ship Structural Analysis for Underwater Explosions", 2nd Cdn Marine Dynamics Conf., UBC, Vancouver, BC, pp 176-184, August 1993.

[5] Slater, J.E. and Erickson, D.A., "Analysis of Fluid-Structure Interaction During Underwater Explosions", 4th Cdn. CFD Conf., Ottawa, ON, oo 287-292, June 1996.

[6] Link, R., Tran, D., Chen, L., Thibault, P.A., Norwood, M.E., and Slater, J.E., "Solution of 2D and 3D Fully Coupled FSI Problems for Compressibel Flows", 3rd Cdn. CFD Conf., Banff, AB, Vol II, pp 71-78, June 1995.

[7] Norwood, M.E., "Development of Computer Codes for Ship Structural Response Analysis to Underwater Explosions", Revised User's Manual to Contracts and W7702-5-R542 and W7707-6-3519, Martec Limited, (being drafted).

[8] Boris, J.P., and Book, D.L., Flux Corrected Transport III: Minimal-Error FCT Algorithms, Journal of Computational Physics, Vol. 20, 1976, pp. 379-431.

[9] Sethian, J.A., Level Set Methods and Fast Marching Methods: Evolving Interfaces in Computational Geometry, Fluid Mechanics, Computer Vision, and Materials Science, Cambridge University Press, 1999.

[10] Benson, D.J., Computational Methods in Lagrangian and Eulerian Hydrocodes, Computer Methods in Applied Mechanics and Engineering, Vol. 99, 1992, pp. 235-394.

[11] J.E.A. John, *Gas Dynamics*, 2nd Edition, Allyn and Bacon Inc., Toronto ON, 109 - 118 (1984).

[12] Hyde, D.W., "CONWEP, Conventional Weapons Effects Program", U.S. Army Engineer Waterways Experiment Station, Vicksburg, MS, 1998.

[13] Slater, J.E. and Rude, G., Loading and Response of Flat Plate Targets From Close-Proximity Underwater Explosions, Proceedings of the 71st Shock and Vibration Symposium, Arlington, VA, Nov., 2000.

[14] Keil, A.H., Response of Ships to Underwater Explosions, Annual Meeting of The Society of Naval Architects and Marine Engineers, Nov 16-17, 1961.

[15] Link, R., Ripley, R., Norwood, M., Josey, T., Donahue, L., and Slater, J., Analysis of the Loading and Response of Flat Plate Targets Subjected to Close-Proximity Underwater Explosions, Proceedings of the 74th Shock and Vibration Symposium, San Diego, CA, Oct., 2003.

[16] Laney, C.B., Computational Gas Dynamics, Cambridge University Press, 1998.

[17] Noh, W.F. and Woodward, P.R., SLIC (Simple Line Interface Method), Lecture Notes in Physics 59. AI Van de Vooren and PJ Zandbergen, editors, pp. 330-340, 1976.

[18] Luton, J.A., and Wardlaw, A.B., Linking Compressible and Incompressible Codes for UNDEX Simulations, Proceedings of the 69th Shock and Vibration Symposium, Minnesota, Oct. 1998.

APPENDIX A

CFD MODELLING OF CLOSE-PROXIMITY UNDER WATER EXPLOSIONS

Presented at 2002 SAVIAC Shock & Vibration Symposium, Newport, RI

CFD Modeling of Close Proximity Underwater Explosions

Rick Link, Frank Lin, David Whitehouse
Martec Limited
Suite 400, 1888 Brunswick St.
Halifax, N.S., Canada, B3J-3J8

John Slater
DRDC Suffield
P.O. Box 4000
Medicine Hat, Alta., Canada, T1A-8K6

A CFD model has been developed for analysis of close proximity underwater explosions to non-responding targets i.e. the bubble jetting problem. An explicit FCT algorithm is adopted for solution of the Euler equations, and a multiple material model assuming mixed properties and pressure equilibrium is used to simulate the interaction between detonation products and water. Calculations performed on deep water models show that the new model is capable of capturing all of the relevant physical phenomena, particularly the large impulse and velocity of the water jet.

INTRODUCTION

The detonation of underwater weapons such as mines and torpedoes is a key factor in the design of naval surface vessels and submarines. Threat assessments have been broadened to include terrorist attacks using devices placed in close proximity to vessels in port. A close proximity underwater explosion subjects the hull structure to an extremely high intensity transient loading. This high transient load is due to the initial shockwave contact, the response induced cavitation formation and closure, and finally the detonation-bubble expansion, collapse, and possible water jet impact. The ability to accurately model such an event requires the ability to simulate all of these phenomena for a range of charge sizes, standoff distances, and target geometries.

Figure 1 shows a schematic of a bubble jet formation on a non-responding target. The detonation products undergo a rapid expansion, and the gas bubble flattens out near the target. The over-expansion of the bubble causes a large low-pressure region in the vicinity of the detonation products-water interface. This low pressure region is eventually repressurized everywhere except for near the target, and the bubble starts to collapse towards the target. Finally, a high-velocity water jet is formed, and the gas bubble forms a toroidal shape. The impulsive load generated from the water jet is significantly larger than that of the initial shock wave, and can cause severe damage.

Traditionally, two phase approaches consisting of a Lagrangian time step and an Eulerian remap step are utilized to solve jetting problems [1]. An alternative approach is described here; a CFD Eulerian solver is utilized, thereby eliminating the need for a Lagrangian step.

METHODOLOGY

In order to provide an accurate numerical simulation of the bubble jet problem, the following must be considered:

1) CFD solver
2) Equations of state
3) Detonation products – water interface

An inviscid CFD solver is adopted here, using the flux corrected transport (FCT) algorithm adopted by Boris [2]. The 2-D axisymmetric equations of mass and momentum (Equations 1 and 2) are solved to obtain material densities, ρ, pressures, p, and radial and horizontal velocities, u and v, respectively:

$$\frac{\partial \rho}{\partial t} + \frac{1}{r}\frac{\partial (r\rho u)}{\partial r} + \frac{\partial (\rho v)}{\partial z} = 0 \qquad [1]$$

$$\frac{\partial (\rho u)}{\partial t} + \frac{1}{r}\frac{\partial (r\rho uu)}{\partial r} - \frac{\partial (\rho uv)}{\partial z} = -\frac{\partial p}{\partial r} \qquad [2a]$$

$$\frac{\partial (\rho v)}{\partial t} + \frac{1}{r}\frac{\partial (r\rho uv)}{\partial r} - \frac{\partial (\rho vv)}{\partial z} = -\frac{\partial p}{\partial z} \qquad [2b]$$

In this formulation, internal energy is not considered, because the material pressure is assumed to be a function of density only.

The gas bubble is assumed to consist of air, and expand isentopically according to Equation 3:

$$p = p_0 \left(\frac{\rho}{\rho_0} \right)^{\gamma} \qquad [3]$$

where

p_0 = 1 atm
ρ_0 = 1.18 kg/m^3
γ = specific heat ratio = 1.4

A modified Tait equation of state is adopted to model the water:

$$p = B\left(\left(\frac{\rho}{\rho_0} \right)^{n} - 1 \right) + p_0 \qquad \frac{\rho}{\rho_0} > 1 \qquad [4]$$

$$p = p_0 \left(\frac{\rho}{\rho_0} \right)^{m} \qquad \frac{\rho}{\rho_0} \le 1 \qquad [5]$$

where

B = 3000 atm
ρ_0 = 1000 kg/ m^3
n = 7.15
m = 21494.0

The exponent, m, is selected so that the slopes of the pressure vs. density curves are equal at atmospheric pressure.

DETONATION PRODUCTS-WATER INTERFACE

Since the bubble jet problem is modeled using discrete computational cells, each cell can have more than one material residing inside of it. Numerous approaches have been used to simulate multi-material problems, including level set methods [3], and mixed cell methods [4]. A mixed cell method assuming equal pressure in all cell constituent materials is adopted here. A mixed density, ρ_{mix}, is used that is a weighted average of the material volume fractions, ϕ_i:

$$\rho_{mix} = \rho_g \phi_g + \rho_w \phi_w \quad [6]$$

The subscripts *g, w* are gas and water constituents, respectively. Conservation of volume dictates that:

$$\phi_g + \phi_w = 1 \quad [7]$$

To obtain the volume fractions, and to maintain a conservative form of the equations, mass fraction tracers, *T*, are used, defined such that:

$$\rho_{mix} T_g = \rho_g \, \phi_g \quad [8]$$

$$\rho_{mix} T_w = \rho_w \, \phi_w \quad [9]$$

and conservation of mass (Equations 6, 8, and 9) dictates that:

$$T_g + T_w = 1 \quad [10]$$

Two extra equations are added to Equations 1 and 2 in order to determine the volume fractions:

$$\frac{\partial(\rho T_i)}{\partial t} + u \frac{\partial(\rho T_i)}{\partial x} = 0 \quad [11]$$

One equation is written each for gas and water.

It remains to determine the pressure in the mixed cell. The equilibrium pressure is determined by using an iterative process; the strategy employed here is as follows:

1) Solve Equations 1, 2, and 11 using FCT to obtain ρ_{mix}, T_g, and T_w.
2) Obtain lower and upper brackets for pressure, p_l and p_u, respectively, shown in Figure 2. These are obtained by assuming a pressure on either side, computing the constituent densities from the equations of state (Equations 3, 4, and 5), computing volume fractions from Equations 8 and 9, and computing the assumed mixed density from Equation 6. In this study, a bracket spacing of a factor of 10 was used between lower and upper pressures.
3) Once bracketed, use false position iteration to solve for the desired level of accuracy on ρ_{mix} shown in Figure 2. This is done by linearly interpolating left and right pressures to obtain a pressure estimate, p_r. From the pressure estimate, a similar procedure to Step 2 is employed to compute an estimate of ρ_{mix}. The bracket, whether left or right, is adjusted, and the iteration proceeds until the desired tolerance level is attained.

RESULTS

To demonstrate the formulation, two problems are analyzed here; a free field bubble pulse, and a bubble jet on a non-responding target. For both problems, the layout is similar; 1 kg of TNT at 1000 m depth is employed. For the free field bubble, one-quarter of the explosive was considered, due to symmetry conditions. An axisymmetric computational grid was used, with an expansion grid used at the uppermost and right sides of the grid. The bubble energy was simulated by using a starting pressure in the gas bubble consistent with the ideal gas equation of state:

$$E = \frac{PV}{\gamma - 1} \qquad [12]$$

where V = gas bubble volume.

Figure 3 is a plot of the water mass fraction tracer and velocity vectors for four times, blue being gas, and red is water; one bubble pulse is shown. It can be seen that the pulse is consistent with observed experimental behavior [5]. The bubble maximum radius and pulse time were calculated as 0.35 m and 7.0 ms, respectively. These compared well with similitude equations obtained from Keil [6] of 0.34 m and 6.6 ms.

An axisymmetric grid with expansion grids similar to those used in the bubble pulse problem were used for the bubble jet. Figure 4 shows the results of the jetting bubble for four times. The bubble initially undergoes a rapid expansion to a maximum radius, and flattens near the target. Eventually, the low pressure region re-pressurizes, and causes the formation of a high velocity water jet. The jet impinges on the bubble, and forms a toroidal shape which dissipates with increasing time. This is consistent with observed experimental results [5].

Figure 5 shows a plot of pressure vs. time at the center of the non-responding target. The results show that there is a large initial pressure from the shockwave that has a relatively small duration. This is followed by a long low-pressure phase and a high pressure-high impulse phase from the jetting bubble. The jetting pressure is significantly longer than that developed by the initial shockwave.

MODEL LIMITATIONS AND FUTURE DIRECTIONS

The bubble model described above produces results that are qualitatively consistent with observed experimental behaviour. However, there were some observed problems with the computational scheme:

1) **FCT scheme** – the FCT algorithm exhibited density oscillations in some problems that resulted in non-physical negative densities. This phenomena has been well documented in the CFD community [7]. A Godunov scheme will now be implemented; this should minimize or eliminate spurious negative densities.
2) **Diffusion** – since the interface was not explicitly tracked to within one cell resolution, several mixed cells resulted. Since air is significantly more compressible than water, the mixed material may be less stiff than what actually exists, and may result in lower pressures and impulses. This will be resolved by including a SLIC or variant interface tracker [8] to maintain a sharp interface.
3) **Cavitation** – bubbles at shallower depths were attempted. However, it was found that the cavitation part of the Tait equation of state may have been too severe; at shallow depths, the bubbles never contracted once the maximum expansion was reached. A floor cavitation pressure similar to that used in Luton and Wardlaw [1] will be used to try and resolve this problem.

Other future directions include deformable (responding) targets, hydrostatic pressure, real explosives, and free surface effects.

REFERENCES

[1] Luton, J.A., and Wardlaw, A.B., Linking Compressible and Incompressible Codes for UNDEX Simulations, Proceedings of the 69th Shock and Vibration Symposium, Minnesota, Oct. 1998.

[2] Boris, J.P., and Book, D.L., Flux Corrected Transport III: Minimal-Error FCT Algorithms, Journal of Computational Physics, Vol. 20, 1976, pp. 379-431.

[3] Sethian, J.A., Level Set Methods and Fast Marching Methods: Evolving Interfaces in Computational Geometry, Fluid Mechanics, Computer Vision, and Materials Science, Cambridge University Press, 1999.

[4] Benson, D.J., Computational Methods in Lagrangian and Eulerian Hydrocodes, Computer Methods in Applied Mechanics and Engineering, Vol. 99, 1992, pp. 235-394.

[5] Slater, J.E. and Rude, G., Loading and Response of Flat Plate Targets From Close-Proximity Underwater Explosions, Proceedings of the 71st Shock and Vibration Symposium, Arlington, VA, Nov., 2000.

[6] Keil, A.H., Response of Ships to Underwater Explosions, Annual Meeting of The Society of Naval Architects and Marine Engineers, Nov 16-17, 1961.

[7] Laney, C.B., Computational Gas Dynamics, Cambridge University Press, 1998.

[8] Noh, W.F. and Woodward, P.R., SLIC (Simple Line Interface Method), Lecture Notes in Physics 59. AI Van de Vooren and PJ Zandbergen, editors, pp. 330-340, 1976.

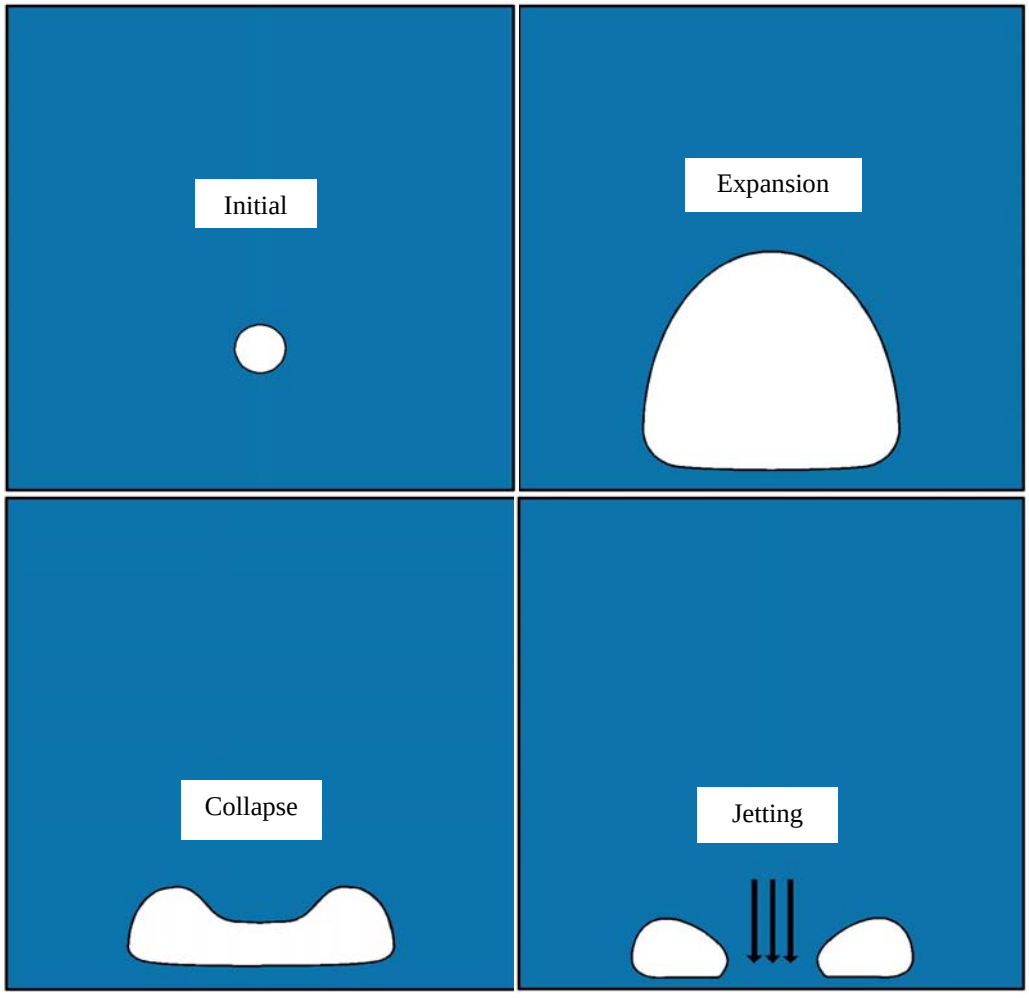


Figure 1: Bubble Jet Schematic (lower boundary is a non-responding target)

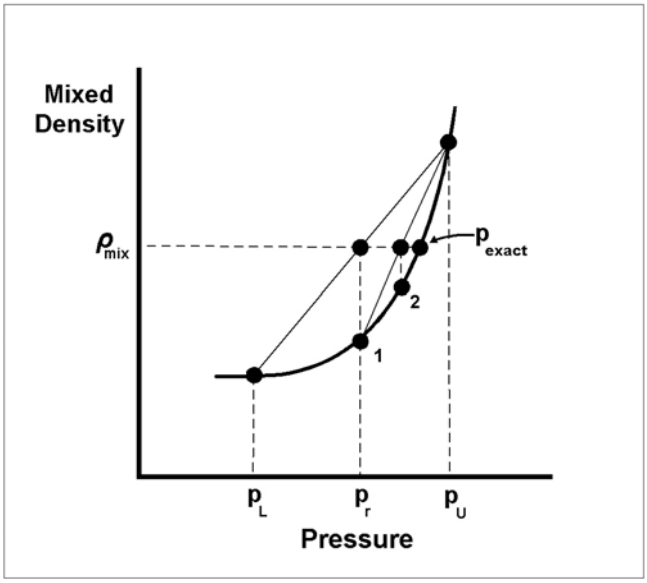


Figure 2: Iterative Method for Pressure Determination

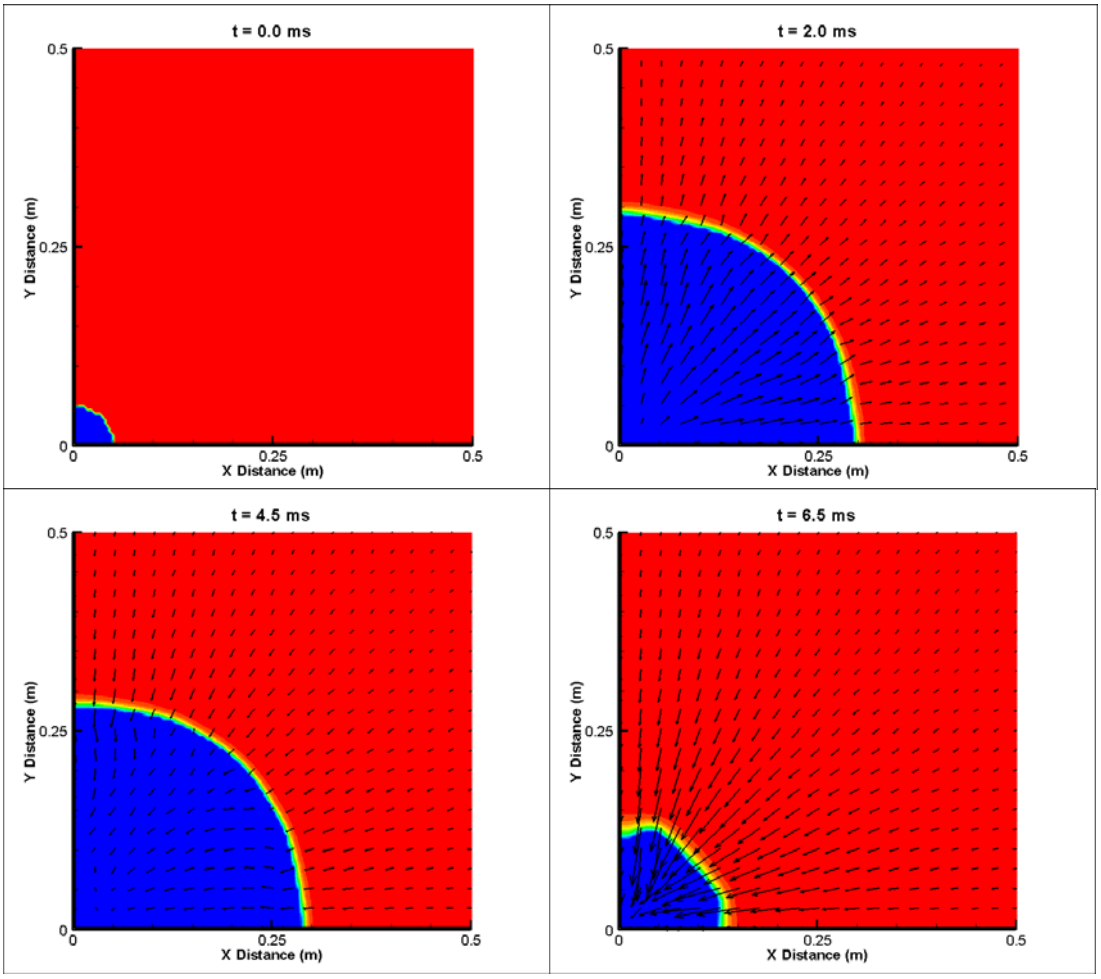


Figure 3: Mass Fraction and Vector Plot - Free-Field Bubble

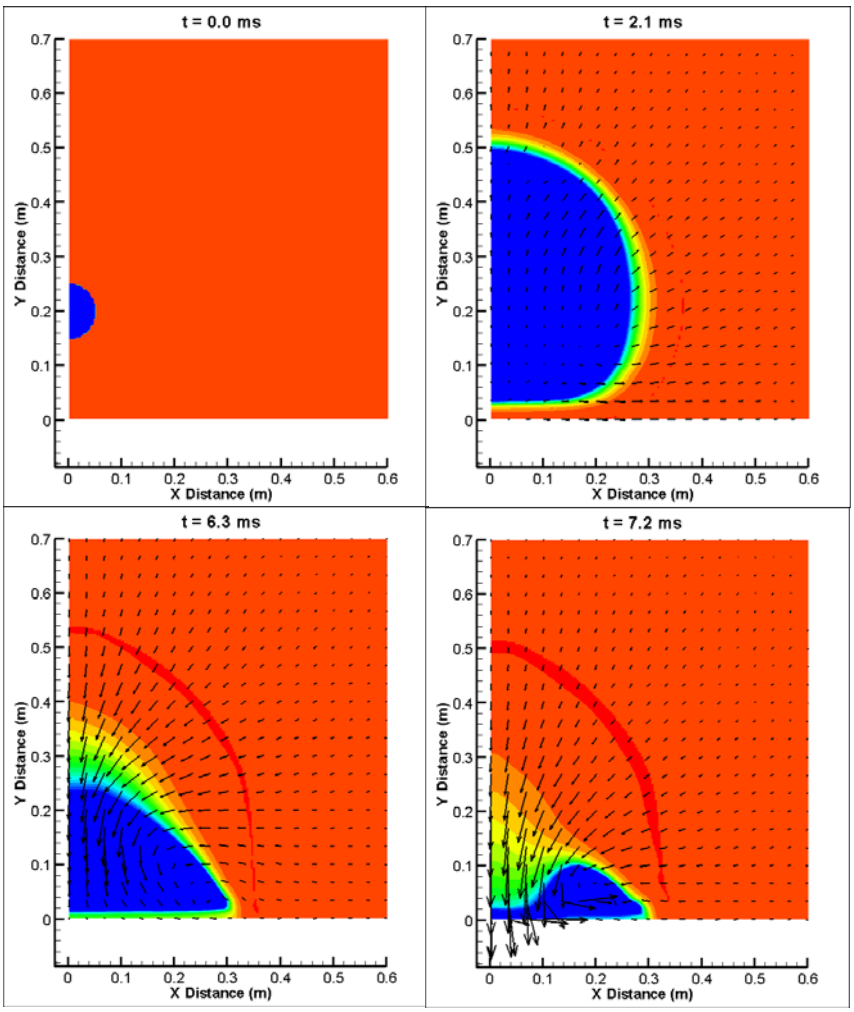


Figure 4: Mass Fraction and Vector Plot - Bubble Jet

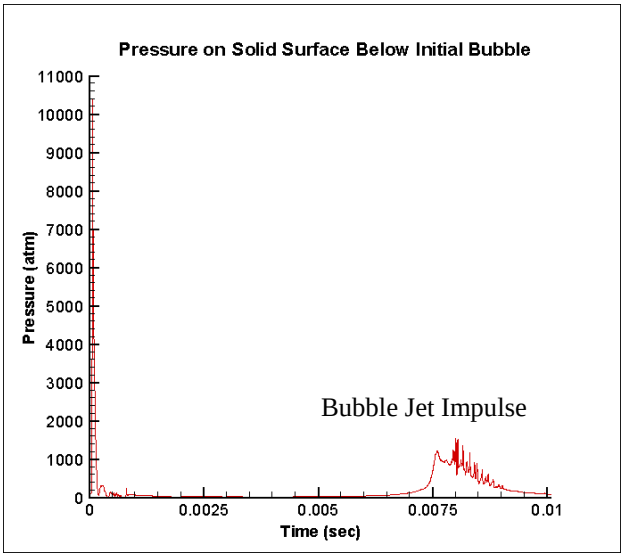


Figure 5: Pressure vs. Time - Bubble Jet

APPENDIX B

TRIDENT-IFSAS CONNECTION EXAMPLE 1: SHIP HULL SUBJECTED TO UNDEX

This tutorial details how to create a hull mesh in Trident and apply an IFSAS load.

SET UP THE PROBLEM

To start a new simulation, begin by creating a new directory (*Hull_Example*) for the problem inside the Trident directory, at the same level as the *exes* directory (using Windows Explorer). Copy the *cfd.inp* file from the *Example_Files* directory into this new directory.

LAUNCH TRIDENT

Now set up the working directory for the problem. *Select Files* → *Working Directory* from the menu bar. The window shown in Figure B.1 will appear.

1) Click the *Browse* button to locate the directory for this problem or type the path to the directory in the edit box field ("Hull_Example")
2) Click the *OK* button when finished

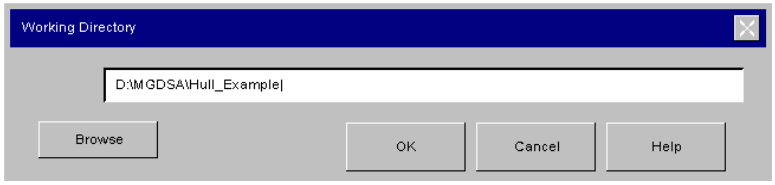


Figure B.1: Working Directory Window

SET UP A NEW PROBLEM DATABASE

To set up a new problem database, begin by selecting *Files* → *Database* from the menu bar. This opens the *Database* window, pictured in Figure B.2.

1) From the *Options* field select *Open New.*
2) In the *Database Prefix* field click *Browse.*
3) Ensure that you are in the working directory for this problem (*Hull_Example*) and enter the database name ("hull_ex"). Click *Open* to close this window and this will place the problem name in the *Database Prefix* edit field.
4) Click *OK* to create the new database.
5) A message will confirm this operation. Click *OK* to close this window.

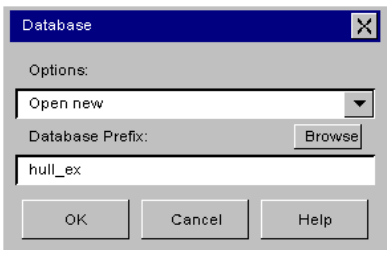


Figure B.2: Database Window

INITIALIZE MODEL UNITS AND ELEMENT PROPERTIES

To initialize the model units and element properties, select *Generate/Modify* → *Properties* from the menu bar.

1) Click *Initialize* from the *Properties* window.
2) Click *Yes* to proceed with the initialization from the popup window.
3) From the *Properties* window click *Model Units* to open the *Model Units* window, shown in Figure B.3.
4) From the *Length Unit* field select *meters*.
5) Leave the *Force Unit* field as *Newtons*.
6) Click *OK* to accept these values and return to the previous window.

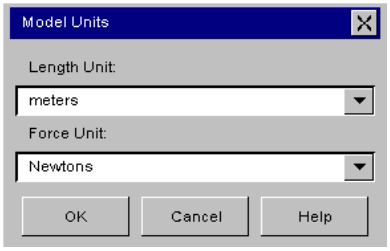


Figure B.3: Model Units Window

Create the element material properties:

1) From the *Properties* window click *Create* to open the *Create Property* window.
2) Click the *Material* button to create a new material.
3) In the material window that appears, enter "1" as the material number and click *OK*.
4) In the *Material Number 1* window (shown in Figure B.4), modify the material name; in the *Name* field type "Steel" and set the other fields to the correct steel properties. Click *OK* to proceed to the next window.
5) Leave the values as their defaults in this window and click *OK*.
6) Click *OK* to the message that indicates the material has been successfully created.
7) Click *Cancel* to the next material window, since a second material is not required

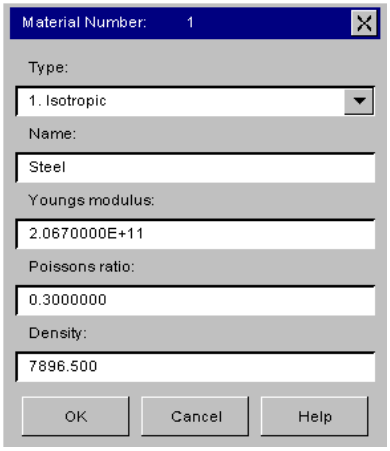


Figure B.4: Material Number 1 Window

Now create the plate properties:

1) From the *Create Property* window click *Plate*.
2) In the next window enter "1" as the plate number and click *OK* to proceed to the next window.
3) In the *Name* field enter *steel_plt*.
4) In the *Material ID* field enter "1".
5) In the *Thickness* field enter "0.00635".
6) Click *OK*.
7) Click *OK* to the message that indicates that the plate has been successfully created.
8) Click *Cancel* to the next material window, since a second plate is not required.
9) Click *Cancel* to close the *Create Property* window.
10) Click *Cancel* to close the *Properties* window.

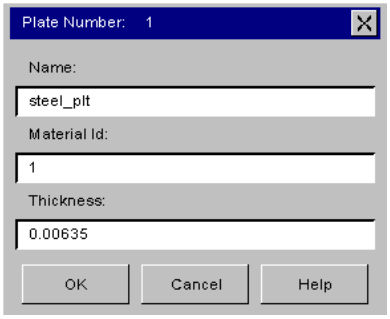


Figure B.5: Plate Number 1 Window

INITIALIZE AND CREATE THE NODES FOR THE HULL

From the *Generate/Modify* menu click *Nodes*.

1) From the *Nodes* window click *Initialize*.
2) Click *Yes* to proceed with the initialization.
3) Click *OK* to the message that indicates the nodes have been initialized.

1) From the *Nodes* window click *Create*. A window similar to the one shown in Figure B.6 will appear.
2) Using the Tables B.1 and B.2, enter the X, Y and Z coordinates for each of the nodes.
3) Click *OK* when the three coordinates have been entered.
4) When you have entered all the coordinates click the *Cancel* button to return to the previous window.
5) Click the *Cancel* button again to close the *Nodes* window.

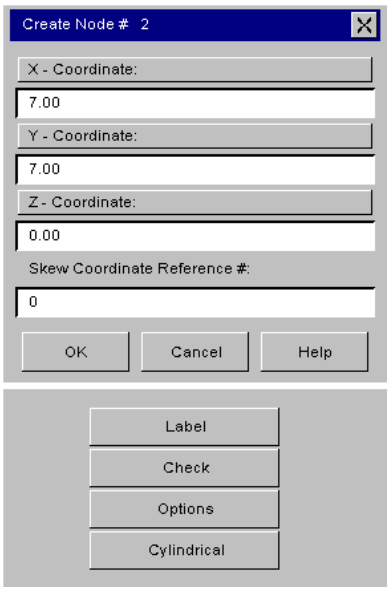


Figure B.6: Create Node Window

Table B.1: Hull Nodes (Part I)

Node	X	Y	Z
1	7.00	7.50	0.00
2	7.00	7.00	0.00
3	7.10	6.50	0.00
4	7.20	6.00	0.00
5	7.40	5.50	0.00
6	7.75	5.00	0.00
7	8.25	4.50	0.00
8	8.75	4.00	0.00
9	9.50	3.70	0.00
10	10.00	3.50	0.00
11	10.50	3.30	0.00
12	11.00	3.00	0.00
13	11.50	3.30	0.00
14	12.00	3.50	0.00
15	12.50	3.70	0.00
16	13.25	4.00	0.00
17	13.75	4.50	0.00
18	14.25	5.00	0.00
19	14.60	5.50	0.00
20	14.80	6.00	0.00
21	14.90	6.50	0.00
22	15.00	7.00	0.00
23	15.00	7.50	0.00
24	7.00	7.50	10.00
25	7.00	7.00	10.00

Table B.2: Hull Nodes (Part II)

Node	X	Y	Z
26	7.10	6.50	10.00
27	7.20	6.00	10.00
28	7.40	5.50	10.00
29	7.75	5.00	10.00
30	8.25	4.50	10.00
31	8.75	4.00	10.00
32	9.50	3.70	10.00
33	10.00	3.50	10.00
34	10.50	3.30	10.00
35	11.00	3.00	10.00
36	11.50	3.30	10.00
37	12.00	3.50	10.00
38	12.50	3.70	10.00
39	13.25	4.00	10.00
40	13.75	4.50	10.00
41	14.25	5.00	10.00
42	14.60	5.50	10.00
43	14.80	6.00	10.00
44	14.90	6.50	10.00
45	15.00	7.00	10.00
46	15.00	7.50	10.00

When all the nodes are entered your workspace should resemble the following:

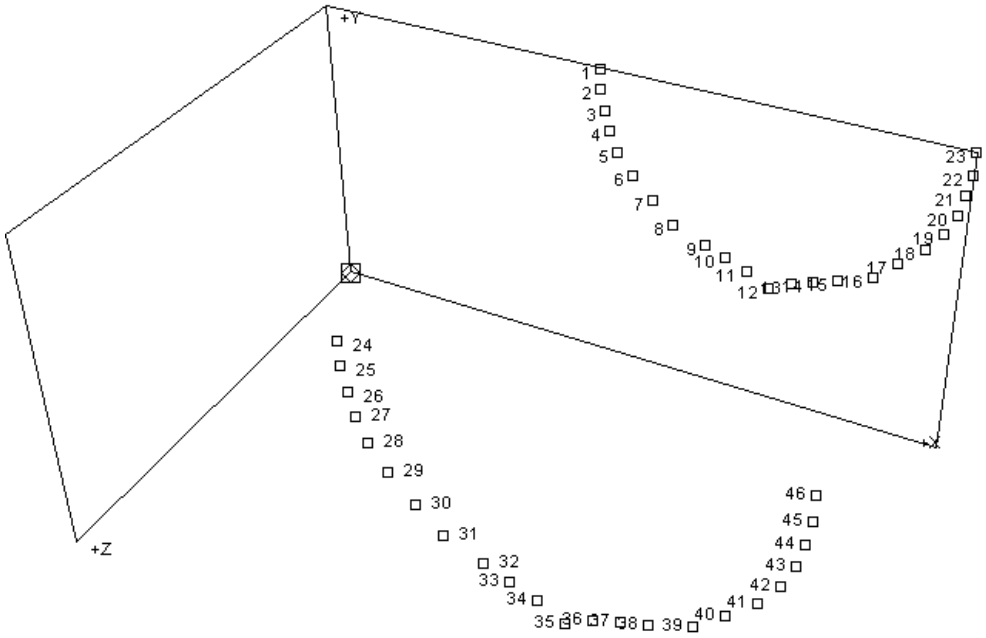


Figure B.7: Workspace with Nodes

CREATE ELEMENTS FORMING THE HULL

To create the hull mesh using the defined nodes, select *Generate/Modify → Mesh* from the menu bar.

1) Click *Plate* from the *Mesh* window to open the *Plate Mesh* window, shown in Figure B.8.
2) From the *Element Type* field select *Quadrilateral Shell.*
3) From the *Number of Edges* field select *Four*.
4) In the *NeIe* fields enter "1-10-1-10".
5) Ensure that the *Plate Property Number* field is set to *1*.
6) Ensure that the *Material Property Number* field is set to *1*.
7) Click *OK* when these materials are entered and use the mouse to select the four nodes which will make up the plate. Select the nodes in order as indicated in the table below (Table B.3).
8) Click OK at the *Biasing Factors* window to accept the default values of *0*. The mesh will then be drawn to the display.
9) If the mesh looks as expected click *Yes* to retain the mesh
10) When finished meshing the hull click *Cancel* to return to the previous window.
11) Click *Cancel* again to close the Mesh window. The finished mesh is shown in Figure B.9.

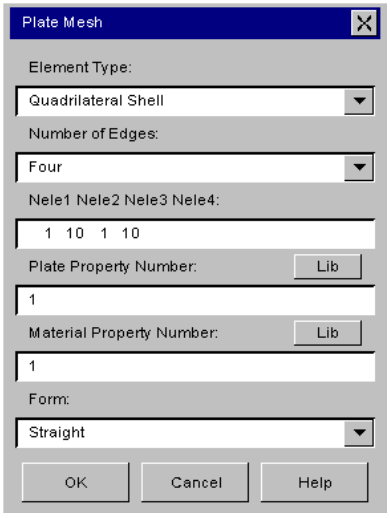


Figure B.8: Plate Mesh Window

Table B.3: Mesh Node Order

Mesh Node Order			
n1	**n2**	**n3**	**n4**
46	45	22	23
45	44	21	22
44	43	20	21
43	42	19	20
42	41	18	19
41	40	17	18
40	39	16	17
39	38	15	16
38	37	14	15
37	36	13	14
36	35	12	13
1	2	25	24
2	3	26	25
3	4	27	26
4	5	28	27
5	6	29	28
6	7	30	29
7	8	31	30
8	9	32	31
9	10	33	32
10	11	34	33
11	12	35	34

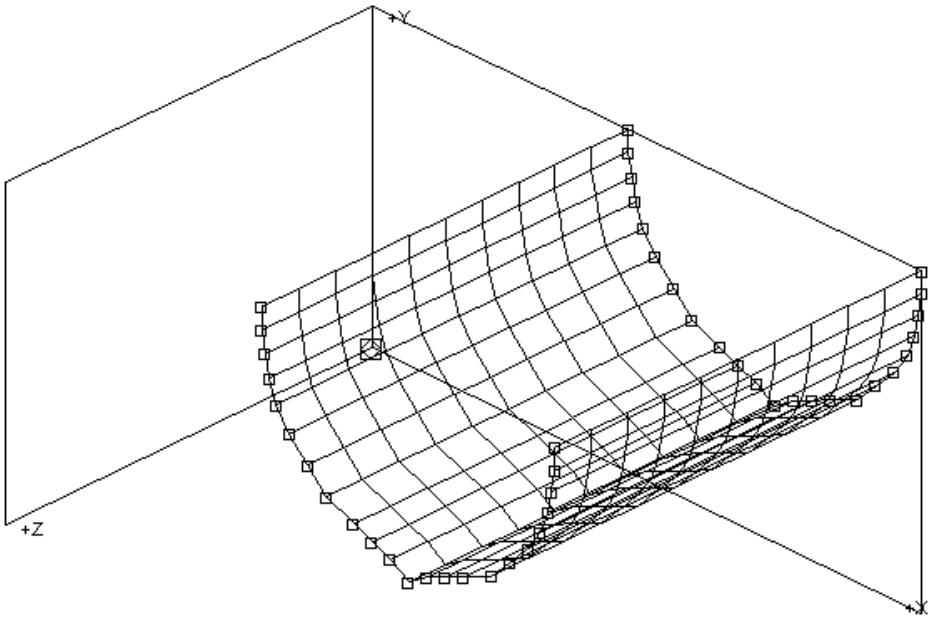


Figure B.9: Workspace with Completed Mesh

VERIFY THE ELEMENT NORMALS

To verify that the element normals are oriented properly, select *Verify* → *Element Normals* from the menu bar.

1) Select *Vectors* from the *Element Normals* window.
2) Ensure that all normals are point outwards, as shown in Figure B.10. If not you will need to reverse the direction of the element's normal.

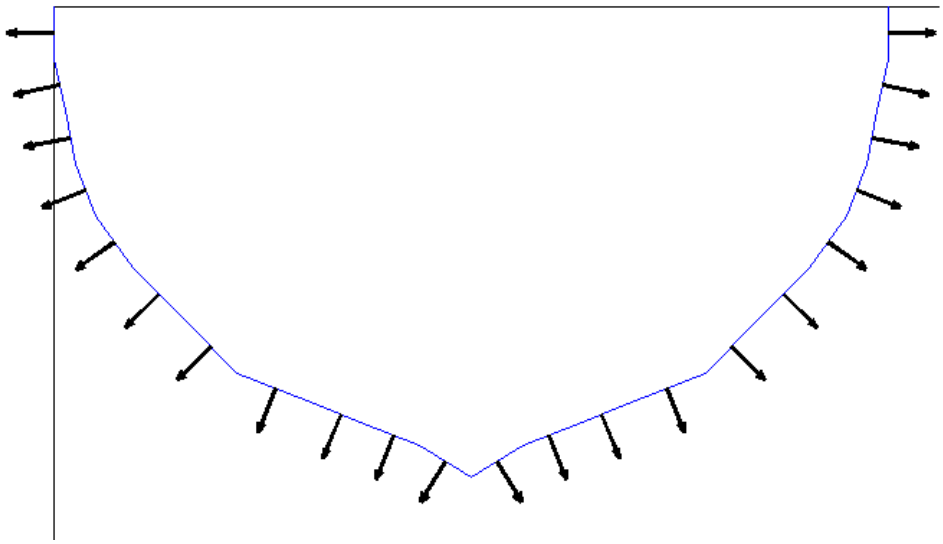


Figure B.10: Element Normals

REMOVE DUPLICATE NODES

To remove duplicate nodes, select *Generate/Modify* → *Nodes* from the menu bar.

1) Click *Equivalence* to open the *Equivalence* window.
2) Select *Radius* and click *OK* to accept the *0.1* meter default radius.
3) Click the *Yes* button to start the equivalence procedure.
4) Click *OK* to close the confirmation window
5) Click *Cancel* to return to the *Nodes* menu
6) Click *Compress* to start the node compressing procedure.
7) Click *OK* to the warning message.
8) Click *Yes* to include the unattached nodes in the compression.
9) Click *OK* to the message which indicates that the nodes have been compressed.
10) Click *Cancel* to close the *Nodes* window

SAVING YOUR WORK

To save the model, select *File* → *Model* from the menu bar.

1) From the *Open/Save* field select *Save.*

2) From the *Model Filename* prefix field enter "hull_ex" or use the *Browse* button to enter the file name prefix.
3) Click ▸ to advance to the next window
4) From the *Save Model Files* window click *All Files*.
5) Click *OK* to save the file at the *Prefix.GOM File* window and leave the subtitle as the default

CREATE THE BOUNDARY CONDITIONS FOR THE HULL

To create the boundary conditions for the hull, begin by selecting *Generate/Modify* → *Boundary Conditions* from the menu bar. To create the boundary conditions for the top edges of the hull:

1) Ensure that *Create* is selected in the *Options* field.
2) Select *Box* in the other field and click *OK* to advance to the *Box* window.
3) Select *Inside* and click *OK*.
4) In *Boundary Conditions* set all fields to *Fixed*.
5) Click the *Z* button on the toolbar at the right to obtain an XY display. This view will aid in selecting the boundaries.
6) Click *OK* from the *Boundary Conditions* window to begin the boundary selection.
7) Using the mouse draw a box around the top portion of the hull, and click the right mouse button when finished (Figure B.11).
8) Click *Yes* to the message to confirm the boundary condition box is correct.
9) Click the *Done* button to return to the menu.

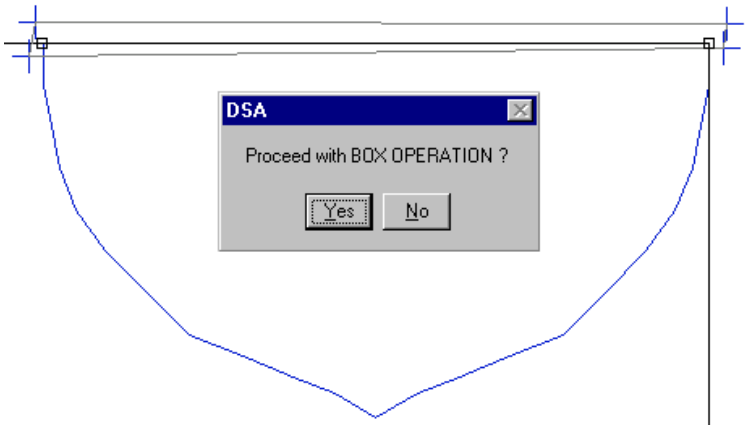


Figure B.11: Top Edge Boundary

To create the boundary conditions for left and right edges of the hull:

1) Click the *Y* button on the toolbar at the right to obtain an XZ display. This view will aid in selecting the other boundaries.
2) Ensure that *Create* is selected in the *Options* field.
3) Select *Box* in the other field and click *OK* to advance to the *Box* window.
4) Select *Inside* and click *OK*.
5) In the *Boundary Conditions* set all fields to *Fixed*.
6) Click *OK* to select the boundaries.

7) Using the mouse draw a box around the left edge portion of the hull, enclosing the top-most nodes (see Figure B.12). Click the right mouse button when finished.
8) Click *Yes* to the message to confirm the boundary condition box is correct
9) Using the mouse draw a box around the right edge portion of the hull, enclosing the bottom-most nodes (see Figure B.12). Click the right mouse button when finished.
10) Click *Yes* to the message to confirm the boundary condition box is correct.
11) Click *Done* when finished with these two boundaries
12) Click the *Cancel* button to close the boundary condition window.

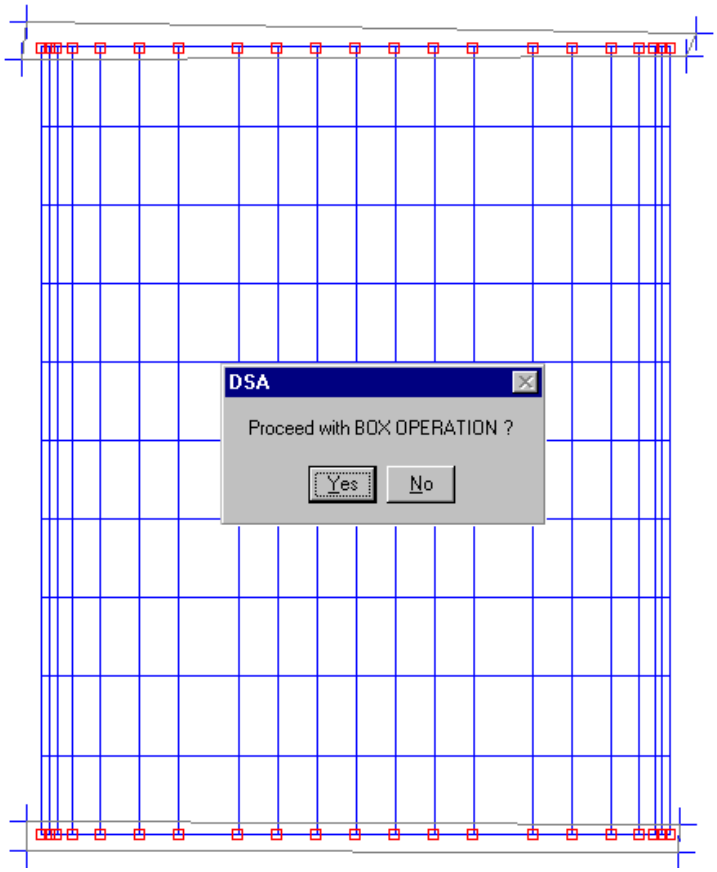


Figure B.12: Left and Right Edge Boundaries

VERIFY BOUNDARY CONDITIONS

To verify that the boundary conditions were created successfully, begin by selecting *Verify* → *Boundary Conditions* from the menu bar.

1) Click *OK* to accept the default values, which are all components and fixed type.
2) The figure below (Figure B.13) indicates the fixed boundaries.

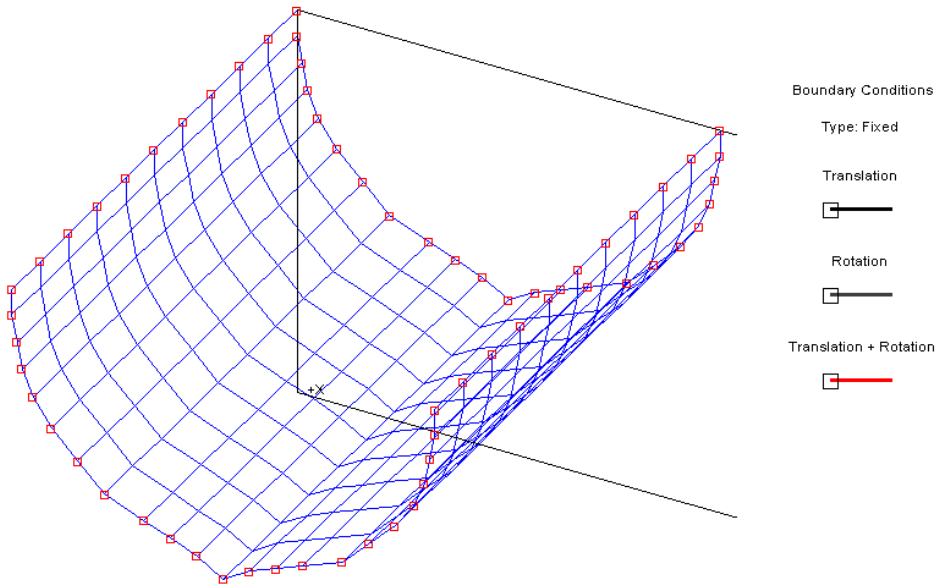


Figure B.13: Boundary Verification

CREATE THE FLUID ELEMENTS

To create the fluid elements, select *Generate/Modify → Fluid Structure* from the menu bar.

1) Select *Structural Element Conversion.*
2) Select *Active Elements* to create fluid elements from all displayed elements.
3) Click the *Yes* button to accept the mesh and close the window.
4) Click *OK* to the message that indicates the number of elements created.

SET THE FLUID ELEMENT PROPERTIES

To set the properties for the fluid elements, select *Generate/Modify → Fluid Elements* from the menu bar.

1) Select *Modify* to open the *Element Types* window.
2) Select *Fluid Elements* from this window to modify the fluid elements only.
3) Select *4 Node Fluid Panel*.
4) From the *Modify* menu select *Box* from the *Single/Box* field and click ▸.
5) Select *Inside* from the *Box* window and click ▸.
6) Click the *All* button to modify all of the fluid elements.
7) Click the *External Air Backed* button to apply this property to the fluid elements. The configuration code should now be set to *1*.
8) Click *OK* to accept these changes.
9) Click ◂ to move to the previous window.
10) Click *Cancel* to close the *Elements* window.

CREATE THE IFSAS NGF FILE

The IFSAS NGF (Neutral Geometry File) is created by Trident by first selecting *Interface* → *IFSAS* from the menu bar.

1) Select *Neutral Geometry File* to create the NGF shape file for IFSAS. This will open the window shown in Figure B.14.
2) In the *File Prefix* field enter the same prefix as before (*hull_ex*)
3) Click the *OK* button to create the file.
4) Click the *OK* button to close the window that indicates the number of elements created.
5) Click the *OK* button to close the window that indicates the file has been created.

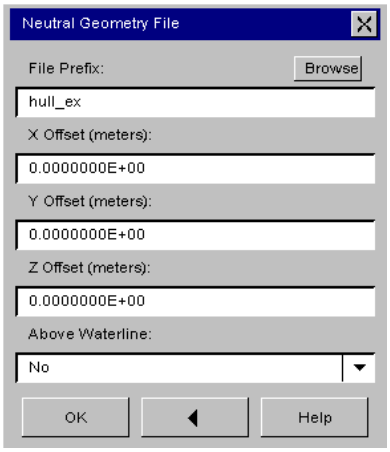


Figure B.14: Neutral Geometry File Window

CREATE THE IFSAS INPUT DATA FILE

The IFSAS input file is created by first selecting *Interface* → *IFSAS* from the menu bar.

1) Select *IFSAS Input Data File* to create the input file for IFSAS. The window shown in Figure B.15 will appear.
2) In the *File Prefix* enter "cfd" as the filenames. Ensure the *NGF File Prefix* is set to *hull_ex*. Click the ▸ button.
3) In the *Grid Geometry Specification* window (Figure B.16) the CFD computational grid is set. Enter the number of cells in X, Y and Z along with the dimension in each direction. These values are shown in the Figure B.16 below. Click the ▸ button.
4) At the *Shape Specification* window (Figure B.17), the centroid of the explosive charge is entered, along with the spherical radius of the charge. Enter the centroid position of the charge, along with the radius and pressure as indicated in Figure B.17 below. Click the *OK* button to accept the changes and create the CFD input file.
5) Click *OK* to the message which indicates the file has been successfully created.
6) Click the ◂ button at the *IFSAS Input Data File* to close the window.
7) Click the *Cancel* button at the *IFSAS* window to close this window. The explosive shape should now appear in the workspace, as shown in Figure B.18.

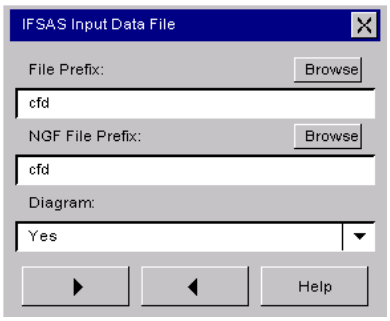


Figure B.15: IFSAS Input File Data Window

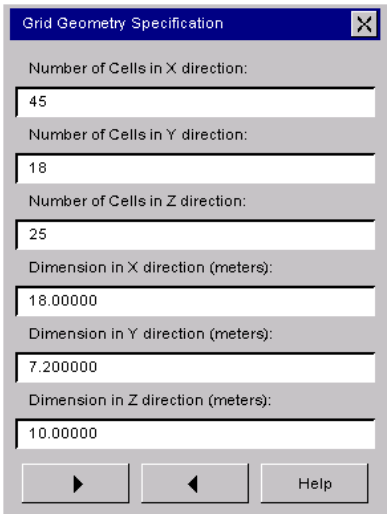


Figure B.16: IFSAS Input File Data Window

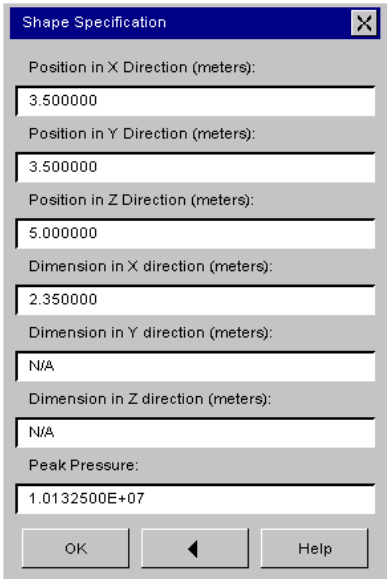


Figure B.17: Shape Specification Window

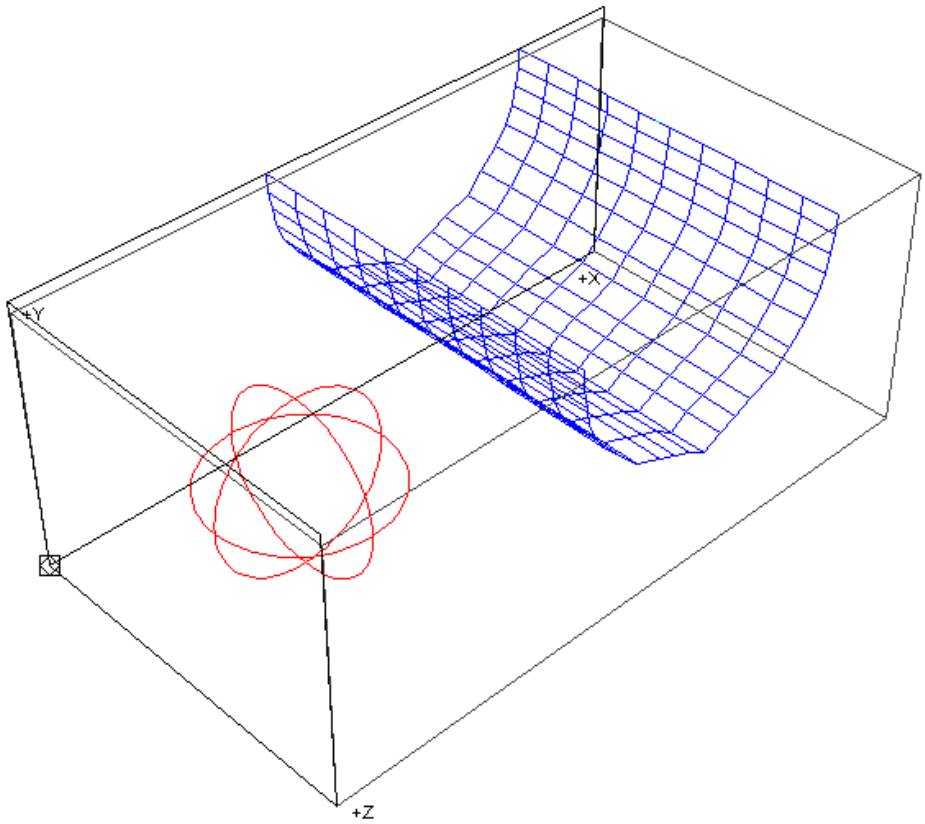


Figure B.18: Domain with Explosive Shape

SAVE ALL FILES

Again, save all files by selecting *Files* → *Model* from the menu bar.

1) From the *Open/Save* field select *Save*.
2) From the *Model Filename Prefix* field enter "hull_ex" or use the *Browse* button to enter this.
3) Click ▸ to advance to the next window.
4) From the *Save Model Files* window click *All Files*.
5) Click *OK* at the *Prefix.GOM File* window and leave the subtitle as the default.
6) Click *Yes* to overwrite the prefix.gom file.
7) Click *OK* at the *Prefix.amd* window and accept the default values.

PREPARE FILES FOR ANALYSIS

Select *Resources* → *FE Solver* from the menu bar to begin preparing the files for the analysis.

1) From the *FE Solver* window (Figure B.19), in the *CFD Analysis* section set the field to *Yes*. Click the *OK* button.
2) In the *FE Solver/CFD* window (Figure B.20), click *Job Information*.
3) In the *Job Information* in the *Job Prefix* field enter "hull_ex" or use the *Browse* button. Click the *OK* button.

4) Click *OK* from the *Prefix Use* file window.
5) Click *OK* at the *Element Results* window.
6) In the *Direct Integration* window leave the defaults, except change the *Lumped Mass File* field to *No* (Figure B.21). Click *OK* to accept the changes
7) In the next *Direct Integration* window (Figure B.22) set the *Number* of final time step to "200". Set the time interval to "0.500E-04". Click the *OK* button.
8) Click the *OK* button to confirm the .use file has been created.

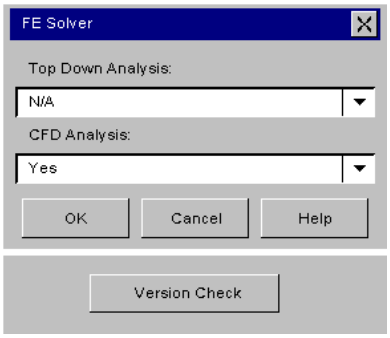


Figure B.19: FE Solver Window

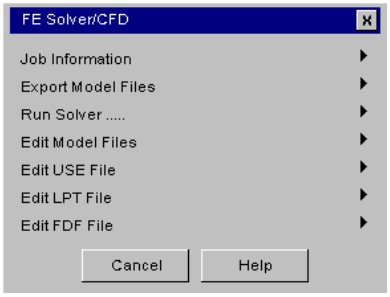


Figure B.20: FE Solver/CFD Window

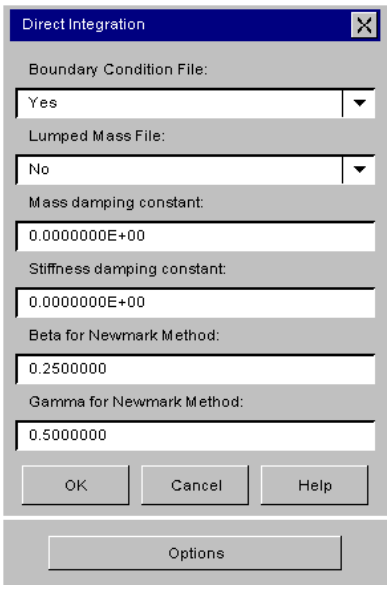


Figure B.21: Direct Integration Window (Part I)

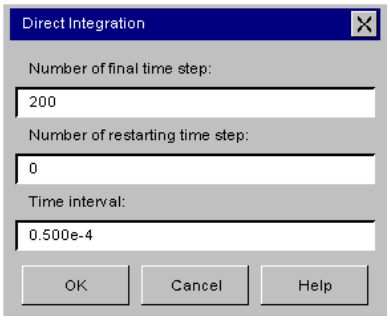


Figure B.22: Direct Integration Window (Part II)

EXECUTE THE ANALYSIS

To execute the analysis:

1) From the *FE Solver/CFD* window select *Run Solver*.
2) Ensure that the *Job Prefix* window (Figure B.23) contains the correct information. If not use the browse button to choose the correct file and directory
3) Click the *OK* button.
4) Click the *Yes* button to start the analysis

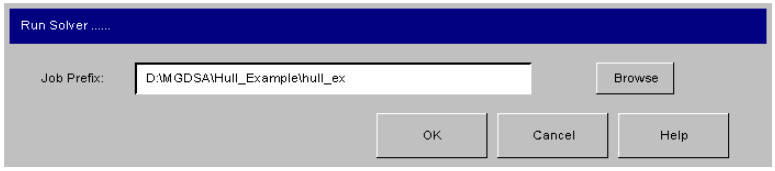


Figure B.23: Run Solver Window

SOLVER COMPLETION

To complete the analysis:

1) Click OK to the window that indicates the solver has completed
2) Click CANCEL to close the FE solver/CFD window

VIEWING THE RESULTS

To view the results, select *Results* → *Deformation* from the menu bar.

1) Select *Displacements* from the *Deformation* window.
2) In the *Dynamic Displacements* window (Figure B.24) enter "–1" in the *Time Step* field and "0.0" in the *Length* field. Click *OK*.
3) The deformation will be displayed. Modify the *Length* field to change the deformation scale if desired.
4) Click *Cancel* to close this window..
5) Click *Cancel* to close the *Deformation* window.

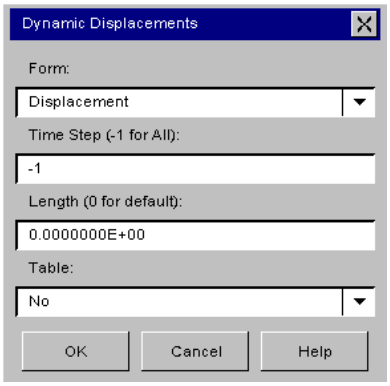


Figure B.24: Dynamic Displacements Window

CREATING AN ANIMATION

To create an animation of the results, begin by selecting *Options* → *Plotting* from the menu bar.

1) Select *Animations* from the *Plotting* window.
2) Select *Create* from the *Display/Create/Delete* field.
3) Click the ▸ button to advance to the next window.
4) Leave the defaults in the Animation window and click ▸.
5) Ensure *Deformations Only* is selected in the *Options* field and click *OK* to start assembling the animation.

To play back the animation:

1) From the *Animation* window select *Display* from the *Display/Create/Delete* field
2) Click the ▸ button to advance to the next window
3) Leave the defaults in this window and click *OK* to start the animation with five cycles.

APPENDIX C

TRIDENT-IFSAS CONNECTION EXAMPLE 2: SHIP SUBJECTED TO UNDEX

In this tutorial, a ship hull mesh will be imported and coupled with an IFSAS II blast.

SET UP THE PROBLEM

To begin the simulation, using Windows Explorer:

1) Create a new directory ("Ship_Example") for the problem inside the Trident directory, at the same level as the exes directory.
2) Copy the *cfd.inp* file from the *Example_Files* directory into this new directory.
3) Copy the *ship_ex.gom* file from the *Example_Files* directory into this new directory.

LAUNCH TRIDENT AND SET UP THE WORKING DIRECTORY

Launch Trident and set up the working directory by first selecting *File* → *Working Directory* from the menu bar.

1) The window shown in Figure C.1 will appear. Click the *Browse* button to locate the directory for this problem, or type the path to the directory in the edit box field ("Ship_Example").
2) Click the *OK* button when finished.

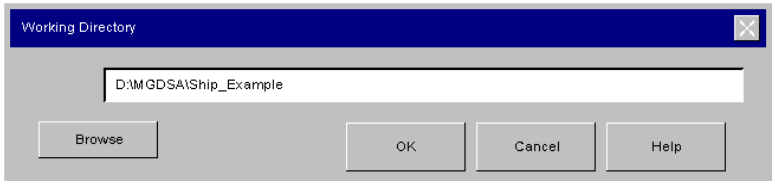


Figure C.1: Working Directory Window

SET UP A NEW PROBLEM DATABASE

To set up a new problem database, select *Files* → *Database* from the menu bar.

1) This opens the *Database* window, shown in Figure C.2. From the *Options* field select *Open New*.
2) In the *Database Prefix* field click *Browse*.
3) Ensure that you are in the working directory for this problem (*Ship_Example*) and enter the database name ("ship_ex"). Click *Open* to close this window and this will place the problem name in the *Database PrefixEedit* field.
4) Click *OK* to create the new database
5) A message will confirm this operation. Click *OK* to close this window.

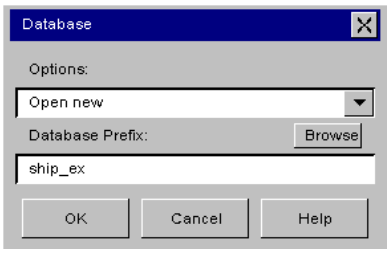


Figure C.2: Database Window

LOAD GEOMETRY FILE

To load the geometry file which includes the structure geometry, material properties and plate properties, select *Files* → *Model* from the menu bar. In this model, the material is steel, with a plate thickness of 0.00635m. The material and plate parameters can be modified if desired.

1) From the *Model* window (Figure C.3) select *Open* from the *Open/Save* field.
2) Click the *Browse* button and locate the *ship_ex.gom* file in the working directory.
3) Click *Open* in the file dialog box.
4) Click the ▸ button.
5) Click *Geometry and Element File* from the *Open Model Files* window.
6) Click *OK* to load the model. The model should now appear in the workspace, as shown in Figure C.5.

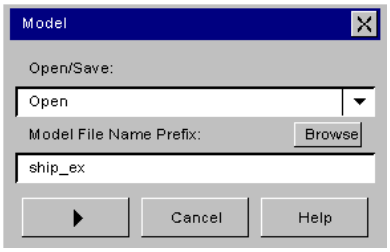


Figure C.3: Model Window

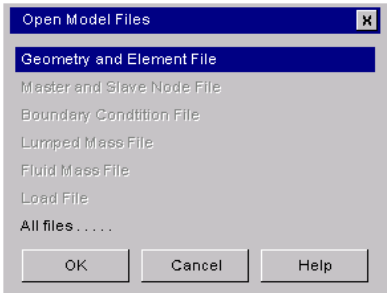


Figure C.4: Open Model Files Window

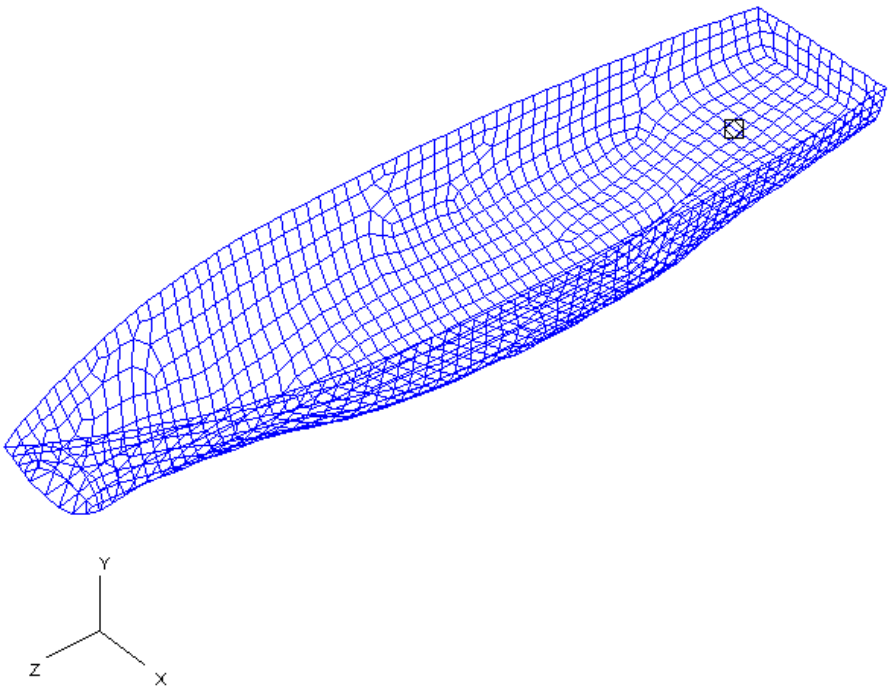


Figure C.5: Loaded Ship Mesh

VERIFY THE ELEMENT NORMALS

The direction of the element normals should now be checked. Select *Verify* → *Element Normals* from the menu bar.

1) Select *Vectors* from the *Element Normals* window.
2) Ensure that all normals point outwards (see Figures C.6 and C.7). If not you will need to reverse the direction of the element's normal.

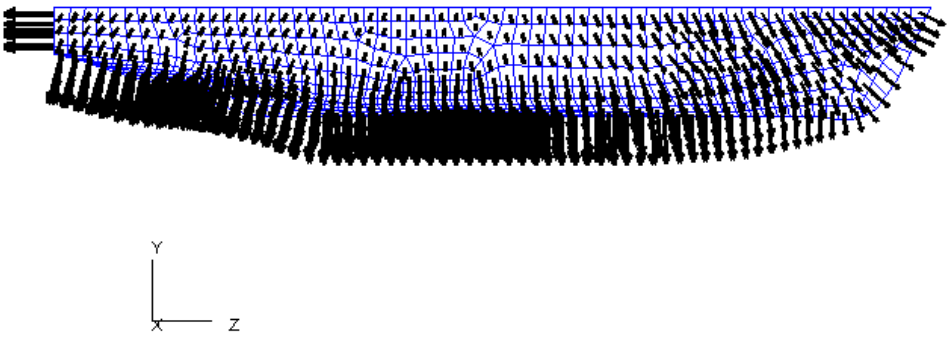


Figure C.6: Ship Mesh with Normals Displayed (Side View)

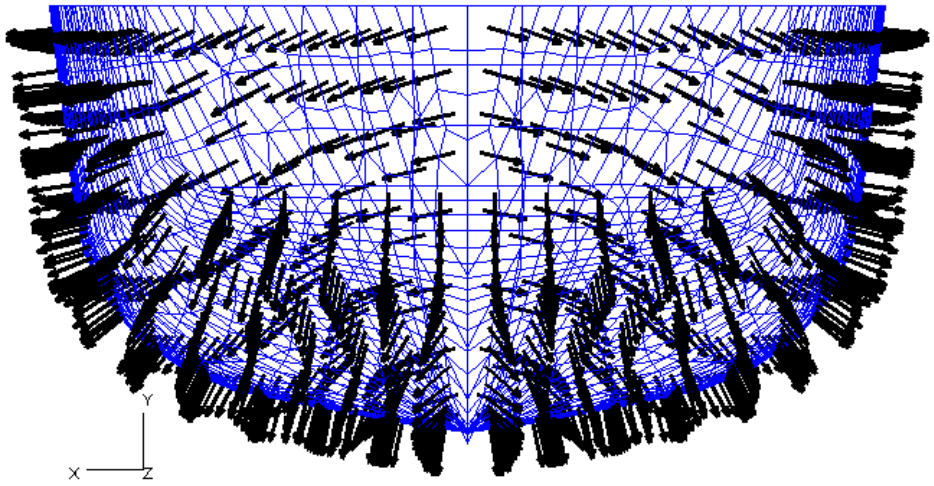


Figure C.7: Ship Mesh with Normals Displayed (End View)

CREATE THE SHIP BOUNDARY CONDITIONS

To create the ship boundary conditions, begin by selecting *Generate/Modify* → *Boundary Conditions* from the menu bar. To create the boundary conditions for the top edges of the ship:

1) Ensure that *Create* is selected in the *Options* field.
2) Select *Box* in the other field and click *OK* to advance to the Box window.
3) Select *Inside* and click *OK.*
4) In the *Boundary Conditions* window set all fields to *Fixed*.
5) Click the *X* button on the toolbar at the right to obtain an YZ display. This view will aid in selecting the boundaries.
6) Click *OK* from the *Boundary Conditions* window to begin the boundary selection.
7) Using the mouse draw a box around the top portion of the ship, and click the right mouse button when finished, as shown in Figure C.8.
8) Click *Yes* to the message to confirm the boundary condition box is correct
9) Click the *Done* button to return to the menu
10) Click the *Cancel* button to close the *Boundary Condition* window.

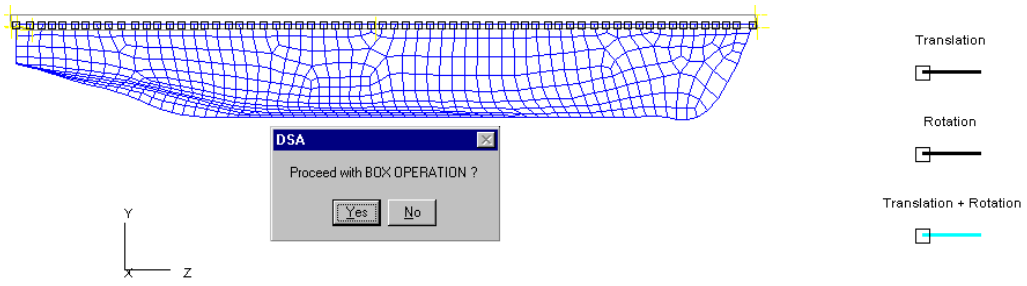


Figure C.8: Ship Model Boundaries

VERIFY THE BOUNDARY CONDITIONS

To verify that the boundary conditions were loaded appropriately, select *Verify* → *Boundary Conditions*. Click *OK* to accept the default values, which are all components and fixed type. The figure below indicates the fixed boundaries.

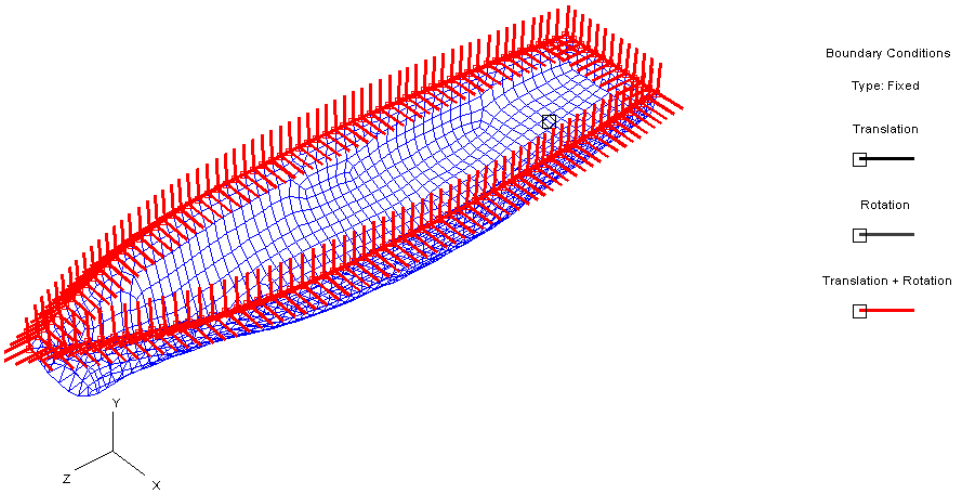


Figure C.9: Ship Model Boundary Verification

CREATE THE FLUID ELEMENTS

To create the surrounding fluid elements, select *Generate/Modify* → *Fluid Structure* from the menu bar.

1) Select *Structural Element Conversion.*
2) Select *Active Elements* to create fluid elements from all displayed elements.
3) Click the *Yes* button to accept the mesh and close the window.
4) Click *OK* to the message that indicates the number of elements created.

SET FLUID ELEMENT PROPERTIES

To set the fluid properties, begin by selecting *Generate/Modify* → *Elements* from the menu bar.

- Select *Modify* to open the *Element Types* window.
- Select *Fluid Elements* from this window to modify the fluid elements only.
- Select *4 Node Fluid Panel.*
- From the *Modify* menu select *Box* from the *Single/Box* field and click ▸.
- Select *Inside* from the *Box* window and click ▸.
- Click the *All* button to modify all of the fluid elements
- Click the *External Air Backed* button to apply this property to the fluid elements. The configuration code should now be set to *1*.
- Click *OK* to accept these changes.
- Click ◂ to move to the previous window.

- Click *Cancel* to close the *Elements* window.

CREATE THE IFSAS NGF FILE

To create the NGF (Neutral Geometry File) of the ship hull for use by IFSAS, select *Interface* → *IFSAS* from the menu bar.

1) Select *Neutral Geometry File* to create the NGF shape file for IFSAS. This will pen the window shown in Figure C.10.
2) In the *File Prefix* field enter the same prefix as before (*ship_ex*).
3) Click the *OK* button to create the file.
4) Click the *OK* button to close the window that indicates the number of elements created.
5) Click the *OK* button to close the window that indicates the file has been created.

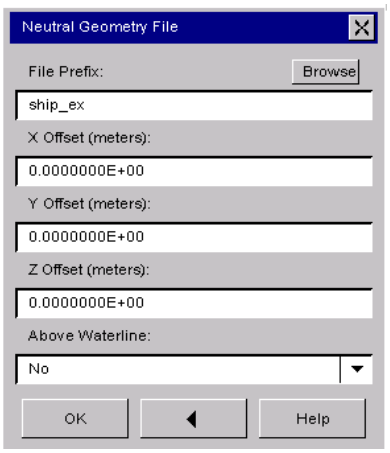


Figure C.10: NGF Window

CREATE THE IFSAS INPUT DATA FILE

The IFSAS input file is created by first selecting *Interface* → *IFSAS* from the menu bar.

1) Select *IFSAS Input Data File* to create the input file for IFSAS. This will open the window shown in Figure C.11.
2) In the *File Prefix* enter "cfd" as the filenames. Ensure the *NGF File Prefix* is set to "ship_ex". Click the ▸ button.
3) In the *Grid Geometry Specification* window (Figure C.12) the CFD computational grid is set. Enter the number of cells in X, Y and Z along with the dimension in each direction. These values are shown in the figure below. Click the ▸ button.
4) At the Shape Specification window (Figure C.13), the centroid of the explosive charge is entered, along with the spherical radius of the charge. Enter the centroid position of the charge, along with the radius and pressure as indicated in Figure C.13 below. Click the OK button to accept the changes and create the CFD input file.
5) Click *OK* to the message which indicates the file has been successfully created.
6) Click the ◂ button at the *IFSAS Input Data File* to close the window.
7) Click the *Cancel* button at the *IFSAS* window to close this window. Figure C.14 shows the workspace with the explosive charge shown.

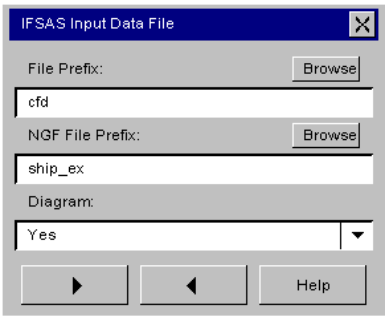


Figure C.11: IFSAS Input File Data Window

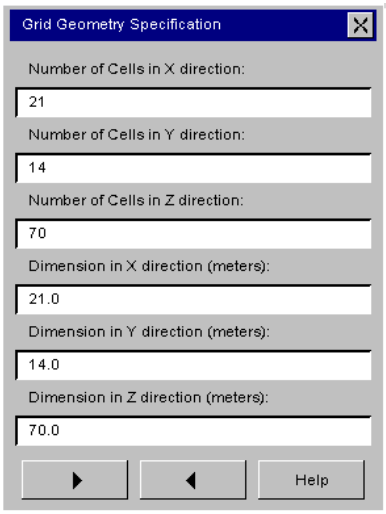


Figure C.12: Grid Geometry Specification Window

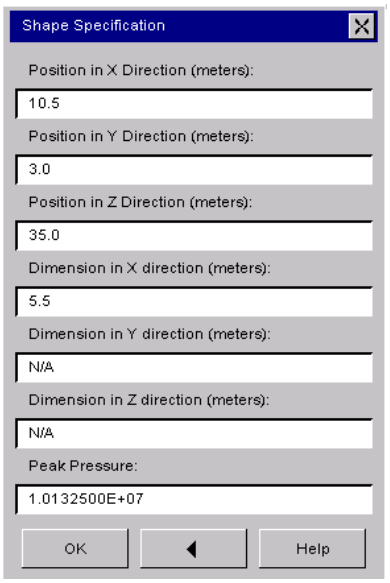


Figure C.13: Shape Specification Window

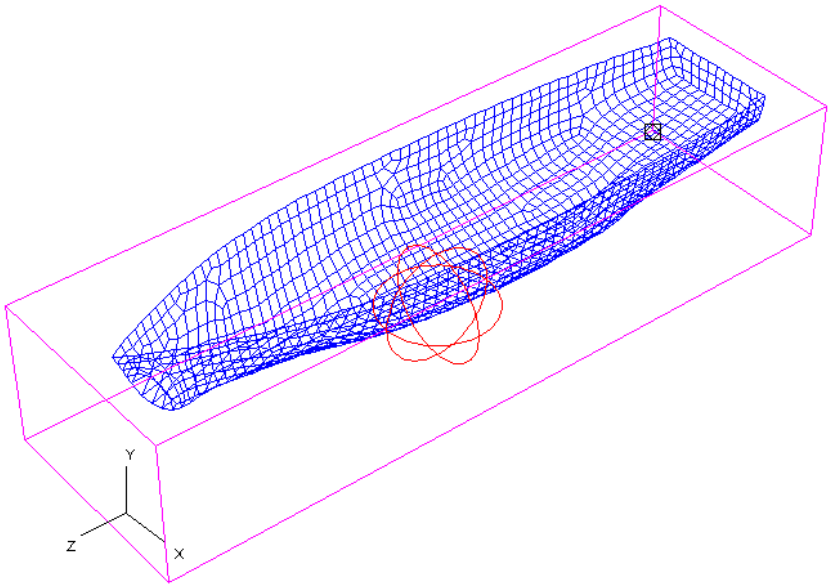


Figure C.14: Domain with Explosive Shape

SAVE ALL FILES

To save the model files, select *Files* → *Model* from the menu bar.

1) From the *Open/Save* field select *Save.*
2) From the *Model Filename Prefix* field enter "ship_ex" or use the *Browse* button to enter this field.
3) Click ▸ to advance to the next window.
4) From the *Save Model Files* window click *All Files.*
5) Click *OK* at the *Prefix.GOM File* window and leave the subtitle as the default.
6) Click *Yes* to overwrite the *prefix.gom* file.
7) Click *OK* at the *Prefix.amd* window and accept the default values.

PREPARE FILES FOR ANALYSIS

Select *Resources* → *FE Solver* from the menu bar to begin preparing the files for the analysis.

1) From the *FE Solver* window (Figure C.15), in the *CFD Analysis* section set the field to *Yes*. Click the *OK* button.
2) In the *FE Solver/CFD* window (Figure C.16), click *Job Information.*
3) In the *Job Information* in the *Job Prefix* field enter "ship_ex" or use the *Browse* button. Click the *OK* button.
4) Click *OK* from the *Prefix Use* file window.
5) Click *OK* at the *Element Results* window.
6) In the *Direct Integration* window leave the defaults, except change the *Lumped Mass File* field to *No* (Figure C.17). Click *OK* to accept the changes.

7) In the next *Direct Integration* window (Figure C.18) set the *Number* of final time step to "200". Set the time interval to "0.500E-04". Click the *OK* button.
8) Click the *OK* button to confirm the .use file has been created.

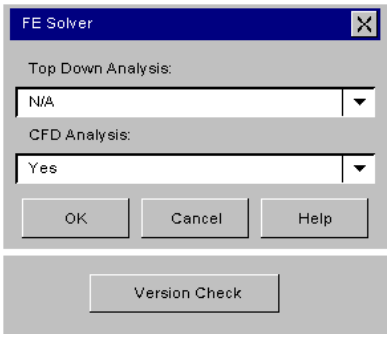


Figure C.15: FE Solver Window

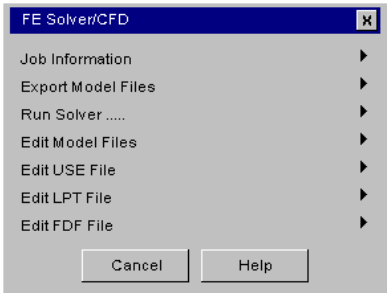


Figure C.16: FE Solver/CFD Window

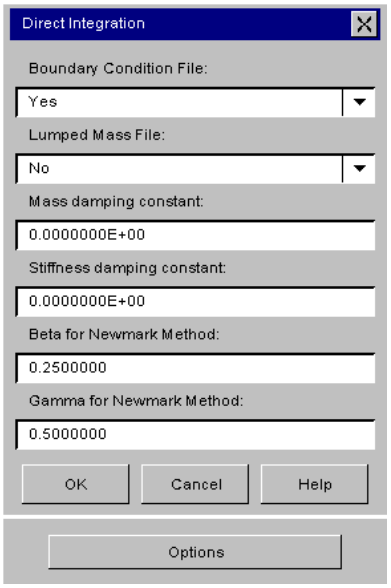


Figure C.17: Direct Integration Window (Part I)

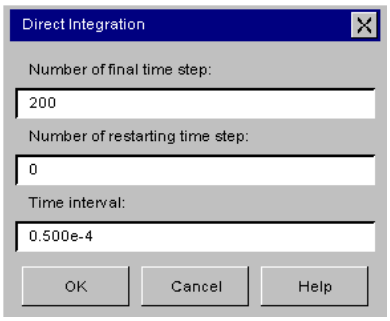


Figure C.18: Direct Integration Window (Part II)

EXECUTE THE ANALYSIS

To execute the analysis:

1) From the *FE Solver/CFD* window select *Run Solver*.
2) Ensure that the *Job Prefix* window (Figure C.19) contains the correct information. If not use the browse button to choose the correct file and directory.
3) Click the *OK* button.
4) Click the *Yes* button to start the analysis.

Figure C.19: Run Solver Window

SOLVER COMPLETION

To complete the analysis:

1) Click OK to the window that indicates the solver has completed.
2) Click CANCEL to close the FE solver/CFD window

VIEWING THE RESULTS

To view the results, select *Results* → *Deformation* from the menu bar.

1) Select *Displacements* from the *Deformation* window.
2) In the *Dynamic Displacements* window (Figure C.20) enter "–1" in the *Time Step* field and "0.0" in the *Length* field. Click *OK*.
3) The deformation will be displayed. Modify the *Length* field to change the deformation scale if desired.
4) Click *Cancel* to close this window.
5) Click *Cancel* to close the *Deformation* window.

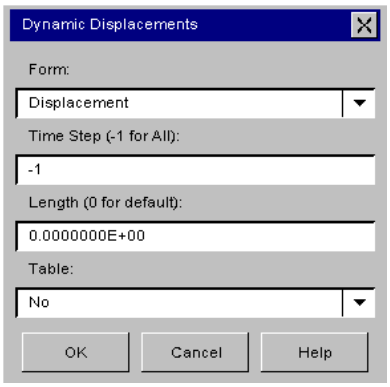


Figure C.20: Dynamic Displacements Window

CREATING AN ANIMATION

To create an animation of the results, begin by selecting *Options* → *Plotting* from the menu bar.

1) Select *Animations* from the *Plotting* window.
2) Select *Create* from the *Display/Create/Delete* field.
3) Click the ▸ button to advance to the next window.
4) Leave the defaults in the Animation window and click ▸.
5) Ensure *Deformations Only* is selected in the *Options* field and click *OK* to start assembling the animation.

To play back the animation:

1) From the *Animation* window select *Display* from the *Display/Create/Delete* field.
2) Click the ▸ button to advance to the next window.
3) Leave the defaults in this window and click *OK* to start the animation with five cycles.

APPENDIX D

TRIDENT-IFSAS CONNECTION EXAMPLE 3: RESPONSE OF STEEL SHELL STRUCTURE TO AIR BLAST

In this tutorial, a the response of a steel shell structure to an internal air blast will be calculated..

SET UP THE PROBLEM

To begin the simulation, using Windows Explorer:

1) Create a new directory ("House_Example") for the problem inside the Trident directory, at the same level as the *exes* directory.
2) Copy the *cfd_house.inp* file from the *Example_Files* directory into this new directory and rename it "cfd.inp".

LAUNCH TRIDENT AND SET UP WORKING DIRECTORY

Launch Trident and set up the working directory for the problem by selecting *Files* → *Working Directory*. This will open the window shown in Figure D.1.

1) Click the *Browse* button to locate the directory for this problem.
2) Or type the path to the directory in the edit box field (*House_Example*).
3) Click the *OK* button when finished.

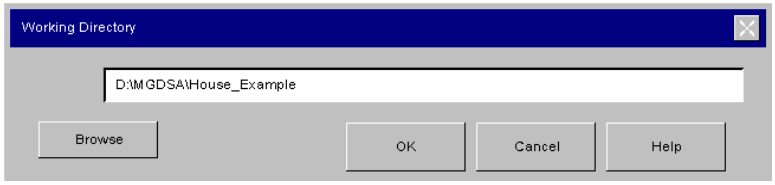


Figure D.1: Working Directory Window

SET UP PROBLEM DATABASE

To set up a new problem database, first select *Files* → *Database* from the menu bar.

1) This opens the *Database* window, pictured in Figure D.2. From the *Options* field select *Open New*.
2) In the *Database Prefix* field click *Browse.*
3) Ensure that you are in the working directory for this problem (*House_Example*) and enter the database name (*house_ex*). Click *Open* to close this window and this will place the problem name in the *Database Prefix* edit field.
4) Click *OK* to create the new database
5) A message will confirm this operation. Click *OK* to close this window.

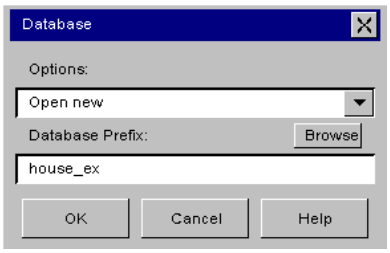


Figure D.2: Database Window

INITIALIZE MODEL UNITS AND ELEMENT PROPERTIES

To initialize the model units and define the element properties, select *Generate/Modify → Properties* from the menu bar.

1) Click *Initialize* from the *Properties* window.
2) Click *Yes* to proceed with the initialization from the popup window.
3) From the *Properties* window click *Model Units* to open the *Model Units* window, shown in Figure D.3.
4) From the *Length Unit* field select *meters*.
5) Leave the *Force Unit* field as *Newtons*.
6) Click *OK* to accept these values and return to the previous window

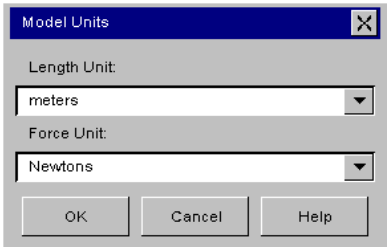


Figure D.3: Model Units Window

Create the element material properties:

1) From the *Properties* window click *Create* to open the *Create Property* window.
2) Click the *Material* button to create a new material.
3) In the *Material* window that appears enter "1" as the material number and click *OK*.
4) In the *Material Number 1* window (Figure D.4), modify the material name; in the *Name* field type *Steel* and set the other fields to the correct steel properties. Click *OK* to proceed to the next window.
5) Leave the values as their defaults in this window and click *OK*.
6) Click OK to the message that indicates the material has been successfully created.
7) Click CANCEL to the next material window, since a second material is not required.

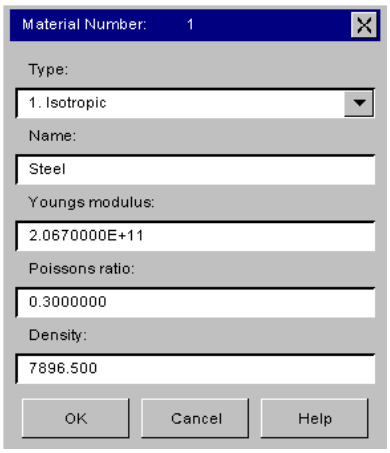


Figure D.4: Model Units Window

Create the plate properties:

1) From the *Create Property* window click *Plate*.
2) In the next window enter "1" as the plate number and click *OK* to proceed to the next window (Figure D.5).
3) In the *Name* field enter "steel_plt".
4) In the *Material ID* field enter "1".
5) In the *Thickness* field enter "0.00635".
6) Click *OK*.
7) Click *OK* to the message that indicates that the plate has been successfully created.
8) Click *Cancel* to the next material window, since a second plate is not required.
9) Click *Cancel* to close the *Create Property* window.
10) Click *Cancel* to close the *Properties* window.

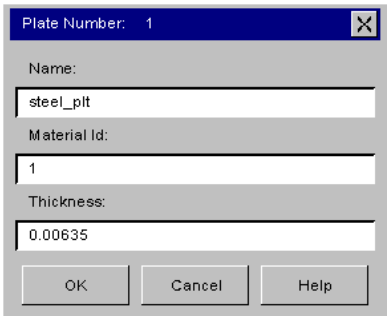


Figure D.5: Model Units Window

INITIALIZE AND CREATE STRUCTURE NODES

To initialize and create nodes for the steel shell structure, first select *Generate/Modify* → *Nodes* from the menu bar.

1) From the *Nodes* window click *Initialize*.

2) Click *Yes* to proceed with the initialization.
3) Click *OK* to the message that indicates the nodes have been initialized.
4) From the *Nodes* window click *Create.* This will open the window shown in Figure D.6.
5) Using the table below (Table C.1), enter the X, Y and Z coordinates for each of the nodes
6) Click *OK* when the three coordinates have been entered.
7) When you have entered all the coordinates click the *Cancel* button to return to the previous window.
8) Click the *Cancel* button again to close the *Nodes* window.

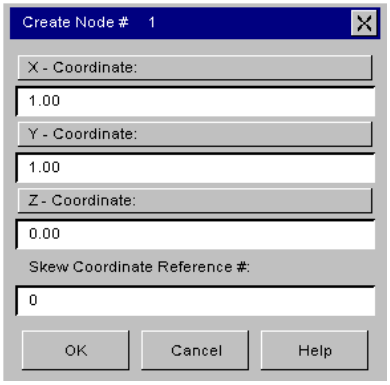


Figure D.6: Create Node Window

Table D.1: Structure Nodes

Node	X	Y	Z
1	1.00	1.00	0.00
2	11.00	1.00	0.00
3	11.00	1.00	4.00
4	1.00	1.00	4.00
5	11.00	11.00	0.00
6	11.00	11.00	4.00
7	1.00	11.00	0.00
8	1.00	11.00	4.00

When all the nodes are entered your workspace should resemble the following:

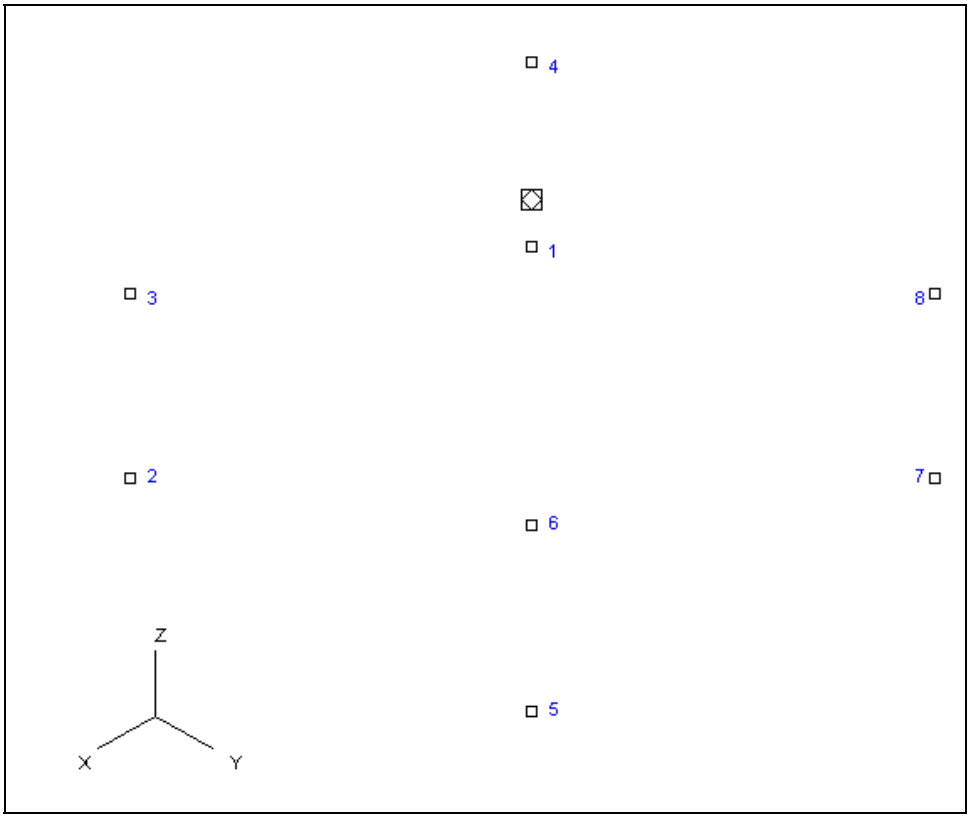


Figure D.7: Workspace with Nodes

CREATE THE STRUCTURE MESH

To create a mesh using the nodes just created, select *Generate/Modify → Mesh* from the menu bar.

1) Click *Plate* from the *Mesh* window to open the *Plate Mesh* window shown in Figure D.8.
2) From the *Element Type* field select *Quadrilateral Shell*.
3) From the *Number of Edges* field select *Four*.
4) In the *NeIe* fields enter "8-20-8-20" for the first four entries in the table below (Table C.2). The first four entries make up the walls of the house. For the last entry, which is the roof, enter "20-20-20-20" in this field.
5) Ensure that the *Plate Property Number* field is set to *1*.
6) Ensure that the *Material Property Number* field is set to *1*.
7) Click *OK* when these materials are entered and use the mouse to select the four nodes which will make up the plate. Select the nodes in order as indicated in the table below (Table C.2).
8) Click *OK* at the Biasing Factors window to accept the default values of *0*. The mesh will then be drawn to the display.
9) If the mesh looks correct click *Yes* to retain the mesh.
10) When finished meshing the house, click *Cancel* to return to the previous window.
11) Click *Cancel* again to close the *Mesh* window

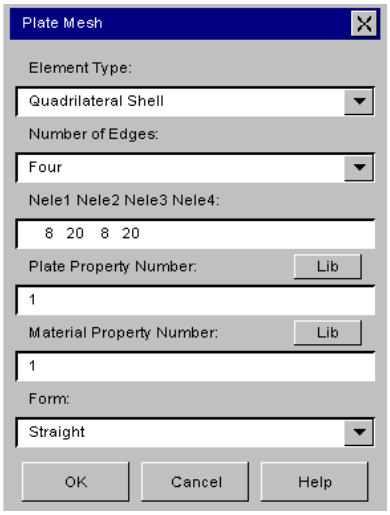


Figure D.8: Plate Mesh Window

Table D.2: Plate Connectivities

Mesh Node Order			
n1	**n2**	**n3**	**n4**
4	1	2	3
3	2	5	6
6	5	7	8
8	7	1	4
8	4	3	6

REMOVE ELEMENTS TO SIMULATE WINDOWS AND DOORS

To remove elements from the mesh, select *Generate/Modify* → *Elements* from the menu bar. First, remove the side windows:

1) From the *Elements* menu select *Delete.*
2) Click the *X* button in order to get a side YZ side view.
3) From the *Delete* window select *Box* from the *Single/Box* field.
4) Click the ▸ button.
5) Select *Inside* from the box window.
6) Click *OK*.
7) Using the mouse create a box around the location of the windows. This box will have a lower corner just above Y=4m, Z=1m and an upper corner just below Y=6m, Z=2.5m. See the diagram below (Figure D.9).
8) Click the *Done* button when finished.

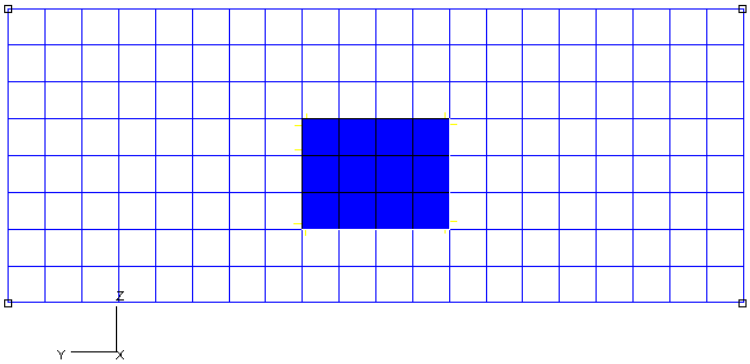


Figure D.9: Elements Selected for Removal (Side View)

Now remove the front windows and door:

- From the *Elements* menu select *Delete.*
- Rotate the model in the view port such that the front face of the house is visible.
- From the *Delete* window select *Box* from the *Single/Box* field.
- Click the ▸ button.
- Select *Inside* from the box window.
- Click *OK*.
- Using the mouse create a box around the location of the two front windows. One box will have a lower corner just above X=1.5m, Z=1m and an upper corner just below X=3m, Z=2.5m. The other box will have a lower corner just above X=7m, Z=1m and an upper corner just below X=8.5m, Z=2.5m. See the Figure D.10.
- Using the mouse create a box around the location of the front door. This box will have a lower corner just above X=4.5m, Z=0m and an upper corner just below X=5.5m, Z=2m. See the Figure D.11.
- Click the *Done* button when finished.

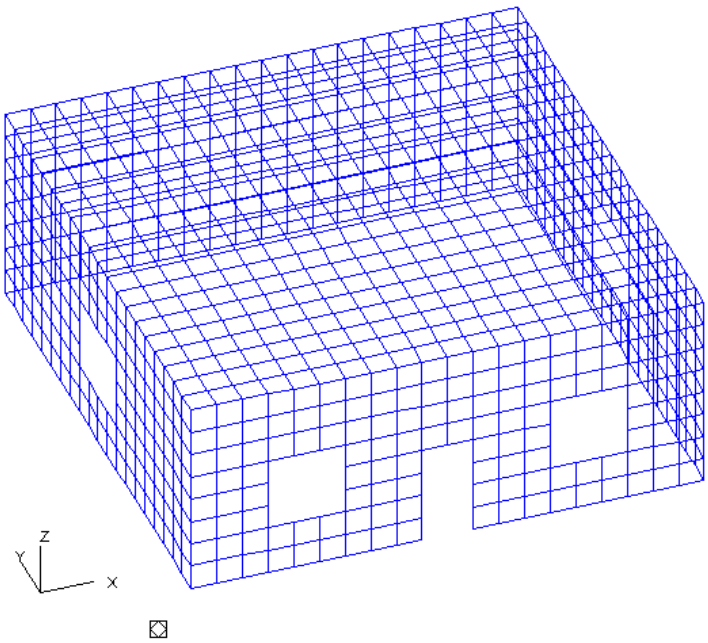


Figure D.10: Structure with Windows

VERIFY THE ELEMENT NORMALS

Now verify that the element normals are oriented correctly. Select *Verify* → *Element Normals* from the menu bar.

1) Select *Vectors* from the *Element Normals* window.
2) Ensure that all normals point outwards (Figures D.11, D.12). If not you will need to reverse the direction of the element's normal.

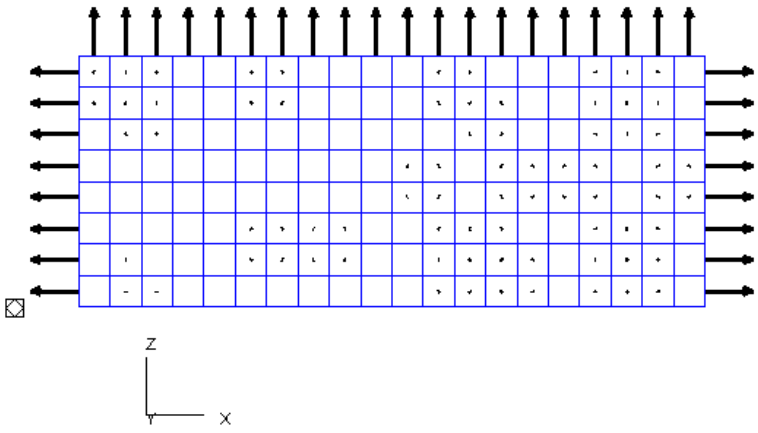


Figure D.11: Correct Normal Directions (Side View)

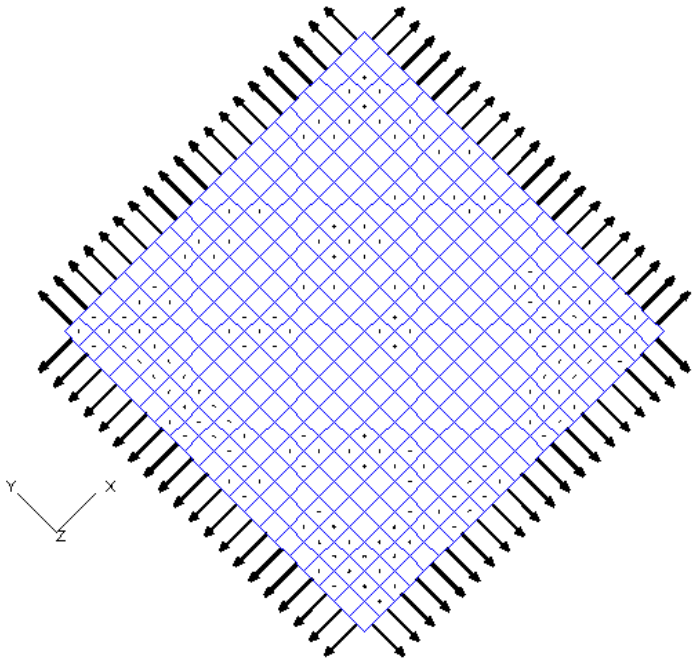


Figure D.12: Correct Normal Directions (Top View)

REMOVE DUPLICATE NODES

To remove duplicate nodes, first select *Generate/Modify* → *Nodes* from the menu bar.

1) Click *Equivalence* to open the *Equivalence* window.
2) Select *Radius* and click *OK* to accept the 0.1 meter default radius.
3) Click the *Yes* button to start the equivalence procedure.
4) Click *OK* to close the confirmation window.
5) Click CANCEL to return to the NODES menu.
6) Click *Compress* to start the node compressing procedure.
7) Click *OK* to the warning message.
8) Click *Yes* to include the unattached nodes in the compression.
9) Click *OK* to the message which indicates that the nodes have been compressed.
10) Click *Cancel* to close the *Nodes* window

SAVING YOUR WORK

To save the model, select *Files* → *Model.*

1) From the *Open/Save* field select *Save*.
2) From the *Model Filename Prefix* field enter "house_ex" or use the *Browse* button to enter the file name prefix.
3) Click ▸ to advance to the next window.
4) From the *Save Model Files* window click *All Files*.
5) Click *OK* to save the file at the *Prefix.GOM File* window and leave the subtitle as the default.

CREATE THE BOUNDARY CONDITIONS

To create the boundary conditions on the bottom surface of the house, begin by selecting *Generate/Modify → Boundary Conditions* from the menu bar.

- Ensure that *Create* is selected in the *Options* field.
- Select *Box* in the other field and click *OK* to advance to the *Box* window.
- Select *Inside* and click *OK*.
- In the *Boundary Conditions* set all fields to *Fixed*.
- Click the *X* button on the toolbar at the right to obtain an YZ display. This view will aid in selecting the boundaries.
- Click *OK* from the *Boundary Conditions* window to begin the boundary selection.
- Using the mouse draw a box around the bottom portion of the house, and click the right mouse button when finished. Box should appear as in Figure D.13.
- Click *Yes* to the message to confirm the boundary condition box is correct
- Click the *Done* button to return to the menu
- Click the *Cancel* button to close the boundary condition window.

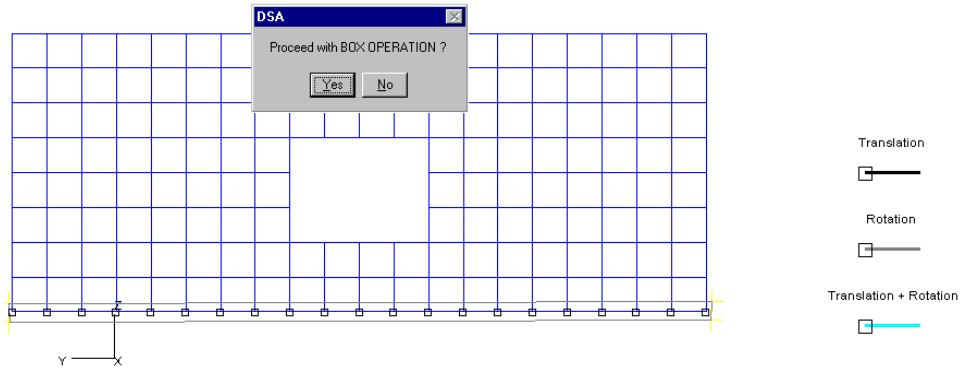


Figure D.13: Boundaries on Lower Surface

VERIFY BOUNDARY CONDITIONS

To verify that the boundary conditions were successfully created, select *Verify → Boundary Conditions* from the menu bar.

1) Click *OK* to accept the default values, which are all components and fixed type.
2) Confirm that the boundaries appear as shown in Figure D.14.

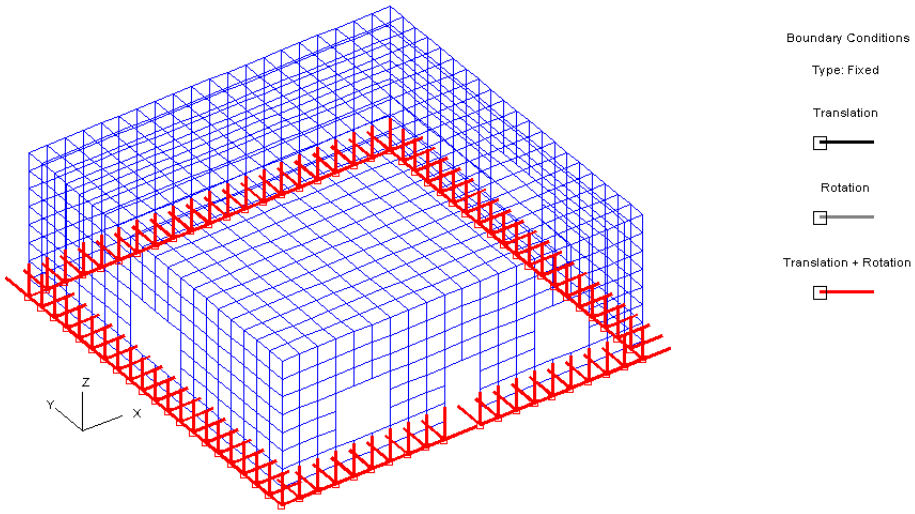


Figure D.14: Structure with Boundary Conditions

CREATE THE FLUID ELEMENTS

Create the fluid elements by first selecting *Generate/Modify* → *Fluid Structure* from the menu bar.

1) Select *Structural Element Conversion.*
2) Select *Active Elements* to create fluid elements from all displayed elements.
3) Click the *Yes* button to accept the mesh and close the window.
4) Click *OK* to the message that indicates the number of elements created.

SET THE FLUID ELEMENT PROPERTIES

Now set the properties for the fluid elements. Select *Generate/Modify* → *Elements* from the menu bar.

1) Select *Modify* to open the *Element Types* window
2) Select *Fluid Elements* from this window to modify the fluid elements only.
3) Select *4 Node Fluid Panel*.
4) From the *Modify* menu select *Box* from the *Single/Box* field and click ▸.
5) Select *Inside* from the *Box* window and click ▸.
6) Click the *All* button to modify all of the fluid elements.
7) Click the *External Air Backed* button to apply this property to the fluid elements. The configuration code should now be set to *1*.
8) Click *OK* to accept these changes.
9) Click ◂ to move to the previous window.
10) Click *Cancel* to close the Elements window.

CREATE THE IFSAS NGF FILE

Create the Neutral Geometry File (NGF) which IFSAS will use to detect the structure in the CFD calculation. Select *Interface* → *IFSAS* from the menu bar.

1) Select *Neutral Geometry File* to create the NGF shape file for IFSAS.
2) In the *File Prefix* field enter the same prefix as before (*house_ex*) (Figure D.15).
3) Click the *OK* button to create the file.
4) Click the *OK* button to close the window that indicates the number of elements created.
5) Click the *OK* button to close the window that indicates the file has been created.
6) Click *Cancel* to close the IFSAS window.
7) Launch a text editor and open the newly created *house_ex.ngf* file. In this file change every occurrence of 32001 to 32002, using the Find/Replace command or its equivalent. Save the file when finished.

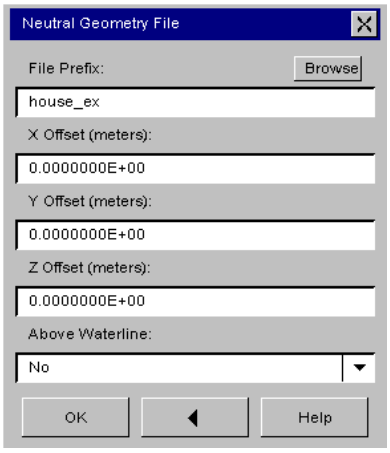


Figure D.15: Neutral Geometry File Window

THE CFD INPUT FILE

The CFD input file was copied from the *Example_Files* directory and placed in the working directory in the opening portions of this example (rename *cfd_house.inp* to "cfd.inp"). The CFD input file has been setup for this example and involves using a TNT explosive profile to place a 10kg TNT charge inside the warehouse structure. For further information on the CFD input file refer to the CFD Input File Manual. The setup of this problem is shown in Figure D.16. The mesh of the building is shown on the CFD grid (30x30x15 cells, 12x12x6 m), along with the explosive profile inside the warehouse.

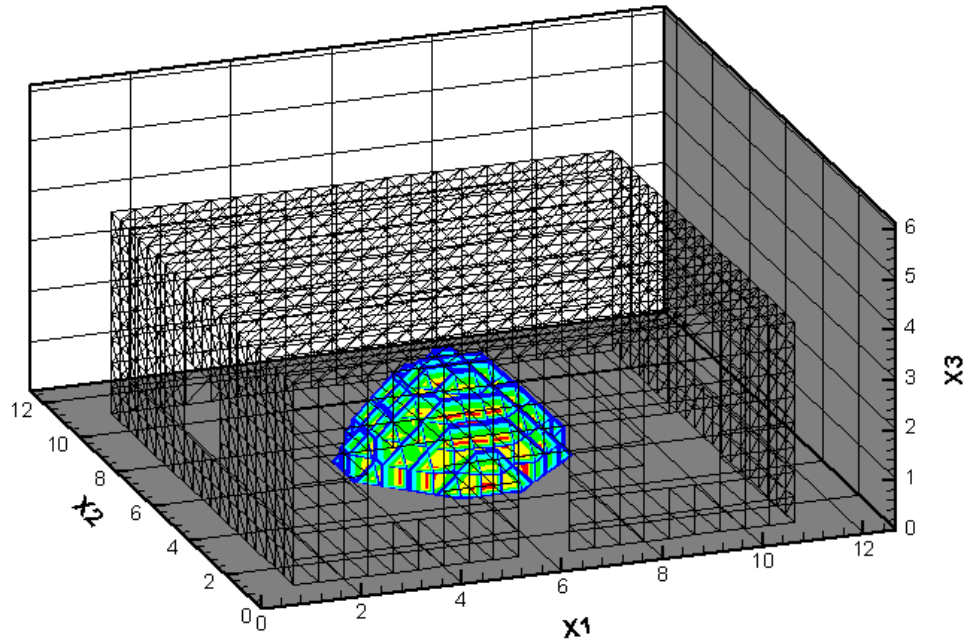


Figure D.16: Structure with Blast

SAVE ALL FILES

To save the model, select *Files* → *Model.*

1) From the *Open/Save* field select *Save*.
2) From the *Model Filename Prefix* field enter "house_ex" or use the *Browse* button to enter the file name prefix.
3) Click ▸ to advance to the next window.
4) From the *Save Model Files* window click *All Files*.
5) Click *OK* to save the file at the *Prefix.GOM File* window and leave the subtitle as the default.
6) Click *Yes* to overwrite the *prefix.gom* file
7) Click *OK* at the *Prefix.amd* window and accept the default values

PREPARE FILES FOR ANALYSIS

Select *Resources* → *FE Solver* from the menu bar to begin preparing file for analysis.

1) From the *FE Solver* window (Figure D.17), in the CFD Analysis set the field to *Yes*. Click the *OK* button.
2) In the *FE Solver/CFD* window click *Job Information.*
3) In the *Job Information* window (Figure D.18), enter "house_ex" *Job Prefix* field or use the *Browse* button. Click the *OK* button.
4) Click *OK* from the *Prefix.use* file window.
5) Click *OK* at the *Element Results* window.
6) In the *Direct Integration* window (Figure D.19) leave the defaults, except change the *Lumped Mass File* field to *No*. Click *OK* to accept the changes.

7) In the next *Direct Integration* window (Figure D.20) set the *Number* of final time step to "80". Set the time interval to "0.100E-03". Click the *OK* button.
8) Click the *OK* button to confirm the .use file has been created.

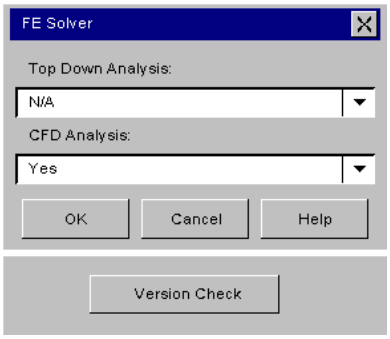


Figure D.17: FE Solver Window

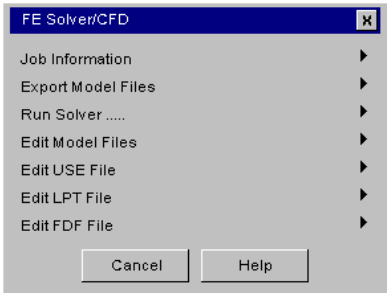


Figure D.18: FE Solver / CFD Window

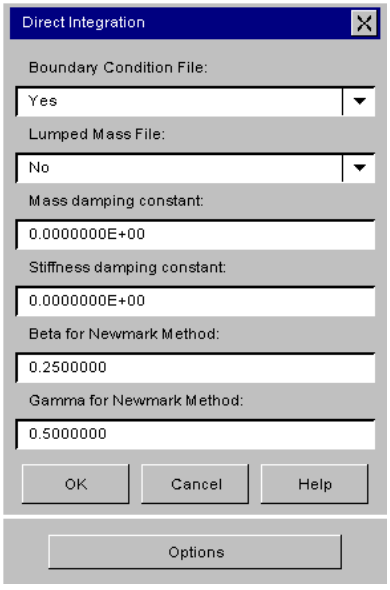


Figure D.19: Direct Integration Window (Part I)

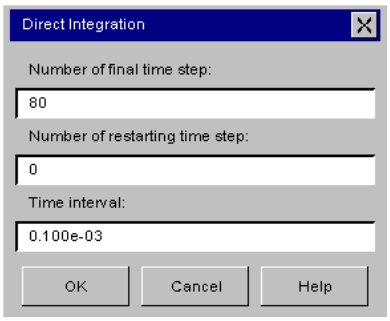


Figure D.20: Direct Integration Window (Part II)

EXECUTE THE ANALYSIS

To execute the analysis:

1) From the *FE Solver/CFD* window select *Run Solver*.
2) Ensure that the *Job Prefix* window (Figure D.21) contains the correct information. If not use the browse button to choose the correct file and directory.
3) Click the *OK* button.
4) Click the *Yes* button to start the analysis.

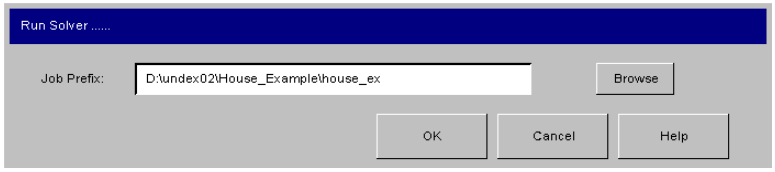


Figure D. 21: Run Solver Window

SOLVER COMPLETION

To complete the analysis:

1) Click OK to the window that indicates the solver has completed.
2) Click CANCEL to close the FE solver/CFD window

VIEWING THE RESULTS

To view the results, select *Results* → *Deformation* from the menu bar.

1) Select *Displacements* from the *Deformation* window.
2) In the *Dynamic Displacements* window (Figure D.22) enter "–1" in the *Time Step* field and "0.0" in the *Length* field. Click *OK*.
3) The deformation will be displayed. Modify the *Length* field to change the deformation scale if desired.
4) Click *Cancel* to close this window.
5) Click *Cancel* to close the *Deformation* window.

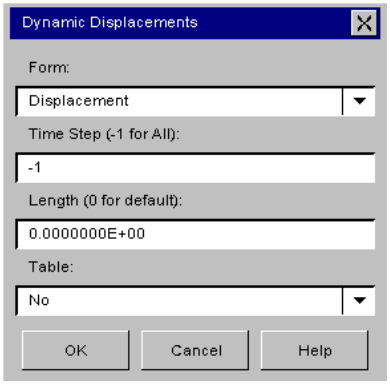


Figure D. 22: Dynamic Displacements Window

CREATING AN ANIMATION

To create an animation of the results, begin by selecting *Options* → *Plotting* from the menu bar.

1) Select *Animations* from the *Plotting* window.
2) Select *Create* from the *Display/Create/Delete* field.
3) Click the ▸ button to advance to the next window.
4) Leave the defaults in the Animation window and click ▸.
5) Ensure *Deformations Only* is selected in the *Options* field and click *OK* to start assembling the animation.

To play back the animation:

1) From the *Animation* window select *Display* from the *Display/Create/Delete* field.
2) Click the ▸ button to advance to the next window.
3) Leave the defaults in this window and click *OK* to start the animation with five cycles.

DOCUMENT CONTROL DATA

(Security classification of title, body of abstract and indexing annotation must be entered when the overall document is classified)

1. ORIGINATOR (the name and address of the organization preparing the document. Organizations for whom the document was prepared, e.g. Centre sponsoring a contractor's report, or tasking agency, are entered in section 8.)

Martec Limited, Suite 400, 1888 Brunswick St., Halifax, NS B3J 3J8

2. SECURITY CLASSIFICATION (overall security classification of the document including special warning terms if applicable).

UNCLASSIFIED

3. TITLE (the complete document title as indicated on the title page. Its classification should be indicated by the appropriate abbreviation (S,C,R or U) in parentheses after the title).

Extension and Validation of UNDEX Analysis Tools

4. AUTHORS (Last name, first name, middle initial. If military, show rank, e.g. Doe, Maj. John E.)

Donahue, L.

5. DATE OF PUBLICATION (month and year of publication of document)

December 2004

6a. NO. OF PAGES (total containing information Include Annexes, Appendices, etc).

86 (approx.)

6b. NO. OF REFS (total cited in document)

8

7. DESCRIPTIVE NOTES (the category of the document, e.g. technical report, technical note or memorandum. If appropriate, enter the type of report, e.g. interim, progress, summary, annual or final. Give the inclusive dates when a specific reporting period is covered).

CONTRACT REPORT

8. SPONSORING ACTIVITY (the name of the department project office or laboratory sponsoring the research and development. Include address).

Defence R&D Canada – Atlantic
PO Box 1012
Dartmouth, NS, Canada B2Y 3Z7

9a. PROJECT OR GRANT NO. (if appropriate, the applicable research and development project or grant number under which the document was written. Please specify whether project or grant).

11gk16

9b. CONTRACT NO. (if appropriate, the applicable number under which the document was written).

W7707-021307/001/HAL

10a ORIGINATOR'S DOCUMENT NUMBER (the official document number by which the document is identified by the originating activity. This number must be unique to this document.)

Martec TR-04-23

10b OTHER DOCUMENT NOs. (Any other numbers which may be assigned this document either by the originator or by the sponsor.)

DRDC Atlantic CR 2004-269

11. DOCUMENT AVAILABILITY (any limitations on further dissemination of the document, other than those imposed by security classification)

(x) Unlimited distribution
( ) Defence departments and defence contractors; further distribution only as approved
( ) Defence departments and Canadian defence contractors; further distribution only as approved
( ) Government departments and agencies; further distribution only as approved
( ) Defence departments; further distribution only as approved
( ) Other (please specify):

12. DOCUMENT ANNOUNCEMENT (any limitation to the bibliographic announcement of this document. This will normally correspond to the Document Availability (11). However, where further distribution (beyond the audience specified in (11) is possible, a wider announcement audience may be selected).

13. ABSTRACT (a brief and factual summary of the document. It may also appear elsewhere in the body of the document itself. It is highly desirable that the abstract of classified documents be unclassified. Each paragraph of the abstract shall begin with an indication of the security classification of the information in the paragraph (unless the document itself is unclassified) represented as (S), (C), (R), or (U). It is not necessary to include here abstracts in both official languages unless the text is bilingual).

Numerous features have been added to both the IFSAS and Trident codes. Both codes were integrated into a single solver for 3D fluid-structure interaction simulations. A multiple material model using a mixed-cell approach was implemented in the IFSAS code, and validated against experimental data. Adaptive grid schemes for underwater shock and bubble analysis, hydrostatic pressure and airwater/seafloor boundaries, underwater explosion profiles, and fluid-backed shapes were also implemented.

The improvements made to the Trident program include upgrading the translators between Trident and USA/CFA Version 5, automating the clipping of the fluid element meshes at the waterline, creating a link to the IFSAS code in the UNDEX shell program, and an extension of capabilities for analysing the response of equipment mounts to UNDEX loading.

14. KEYWORDS, DESCRIPTORS or IDENTIFIERS (technically meaningful terms or short phrases that characterize a document and could be helpful in cataloguing the document. They should be selected so that no security classification is required. Identifiers, such as equipment model designation, trade name, military project code name, geographic location may also be included. If possible keywords should be selected from a published thesaurus. e.g. Thesaurus of Engineering and Scientific Terms (TEST) and that thesaurus-identified. If it not possible to select indexing terms which are Unclassified, the classification of each should be indicated as with the title).

underwater explosion
UNDEX
computational fluid dynamics
fluid-structure interaction
shock wave
gas bubble dynamics
finite element analysis

This page intentionally left blank.

Defence R&D Canada

Canada's leader in defence and national security R&D

R & D pour la défense Canada

Chef de file au Canada en R & D pour la défense et la sécurité nationale

www.drdc-rddc.gc.ca